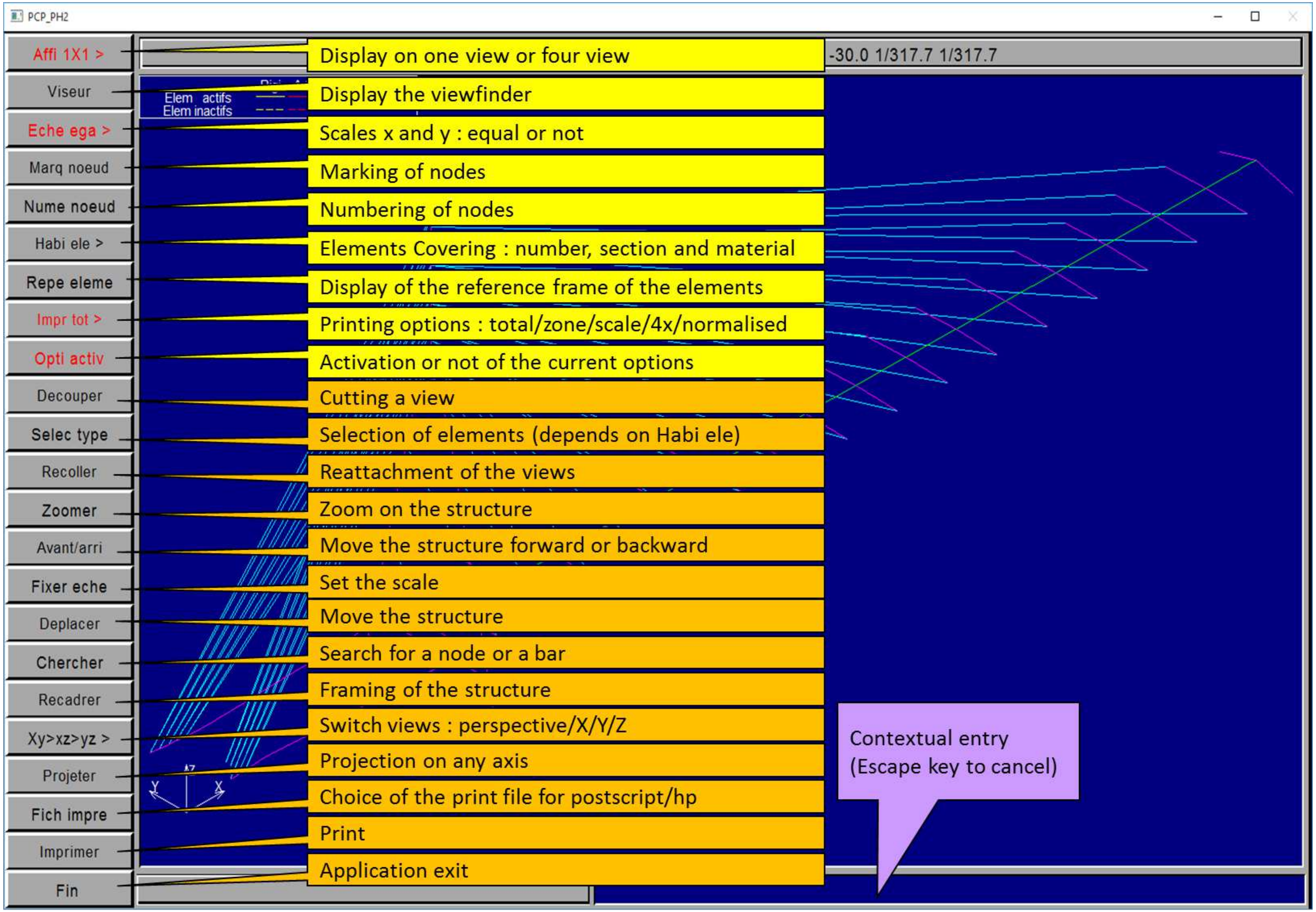


Affi 1X1 >	Display on one view or four view
Viseur	Display the viewfinder
Eche ega >	Scales x and y : equal or not
Marq noeud	Marking of nodes
Nume noeud	Numbering of nodes
Habi ele >	Elements Covering : number, section and material
Repe eleme	Display of the reference frame of the elements
Impr tot >	Printing options : total/zone/scale/4x/normalised
Opti activ	Activation or not of the current options
Decouper	Cutting a view
Selec type	Selection of elements (depends on Habi ele)
Recoller	Reattachment of the views
Zoomer	Zoom on the structure
Avant/arri	Move the structure forward or backward
Fixer eche	Set the scale
Deplacer	Move the structure
Chercher	Search for a node or a bar
Recadrer	Framing of the structure
Xy>xz>yz >	Switch views : perspective/X/Y/Z
Projeter	Projection on any axis
Fich impre	Choice of the print file for postscript/hp
Imprimer	Print
Fin	Application exit

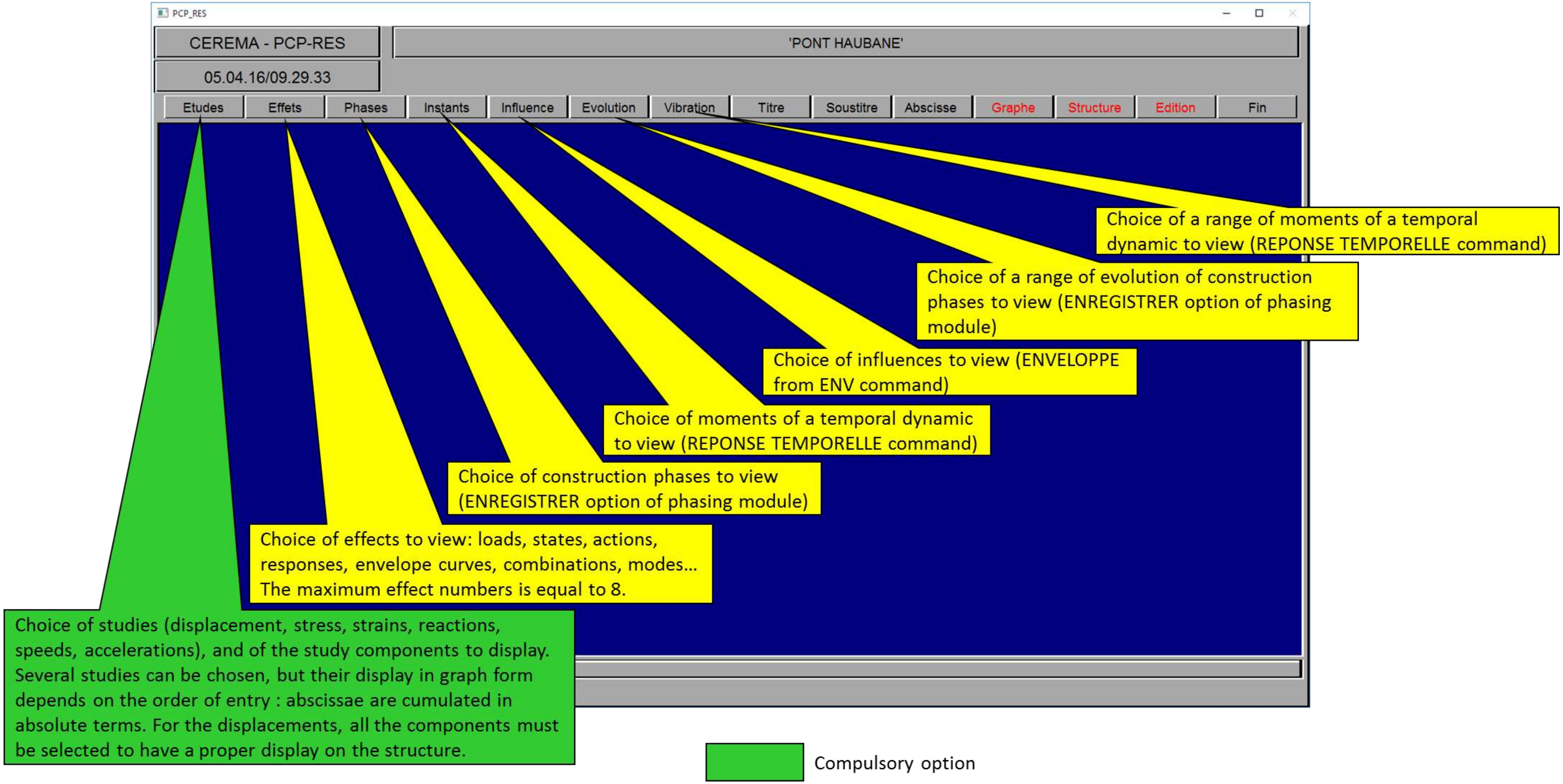
Contextual entry
(Escape key to cancel)

🟨 Options

🟧 Actions

🟪 Entries

Results : choice of data to display



Choice of a range of moments of a temporal dynamic to view (REPONSE TEMPORELLE command)

Choice of a range of evolution of construction phases to view (ENREGISTRER option of phasing module)

Choice of influences to view (ENVELOPPE from ENV command)

Choice of moments of a temporal dynamic to view (REPONSE TEMPORELLE command)

Choice of construction phases to view (ENREGISTRER option of phasing module)

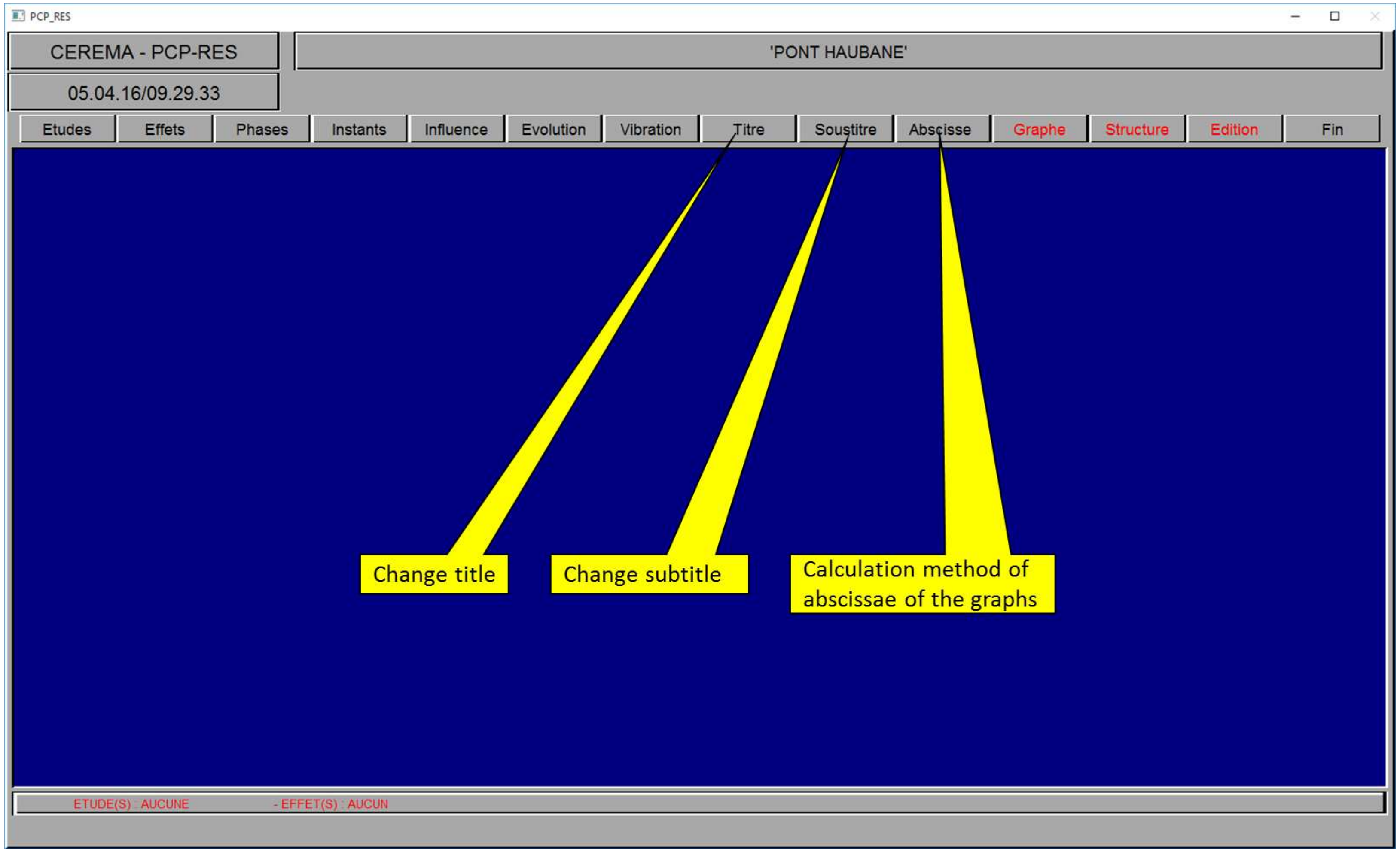
Choice of effects to view: loads, states, actions, responses, envelope curves, combinations, modes… The maximum effect numbers is equal to 8.

Choice of studies (displacement, stress, strains, reactions, speeds, accelerations), and of the study components to display. Several studies can be chosen, but their display in graph form depends on the order of entry : abscissae are cumulated in absolute terms. For the displacements, all the components must be selected to have a proper display on the structure.

🟩 Compulsory option

🟨 Exclusive options

Results : additional options

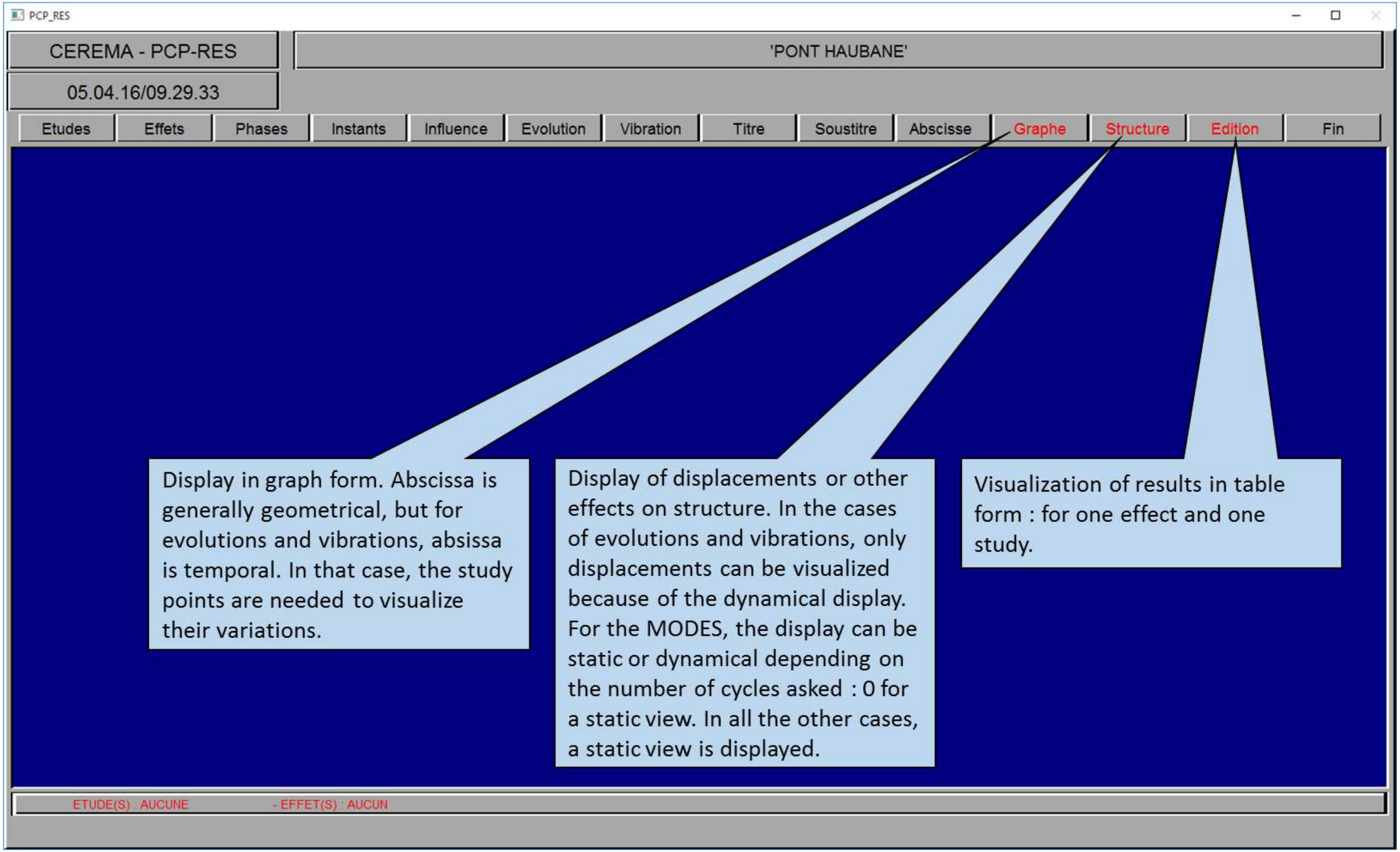


Change title

Change subtitle

Calculation method of abscissae of the graphs

Results : visualization actions

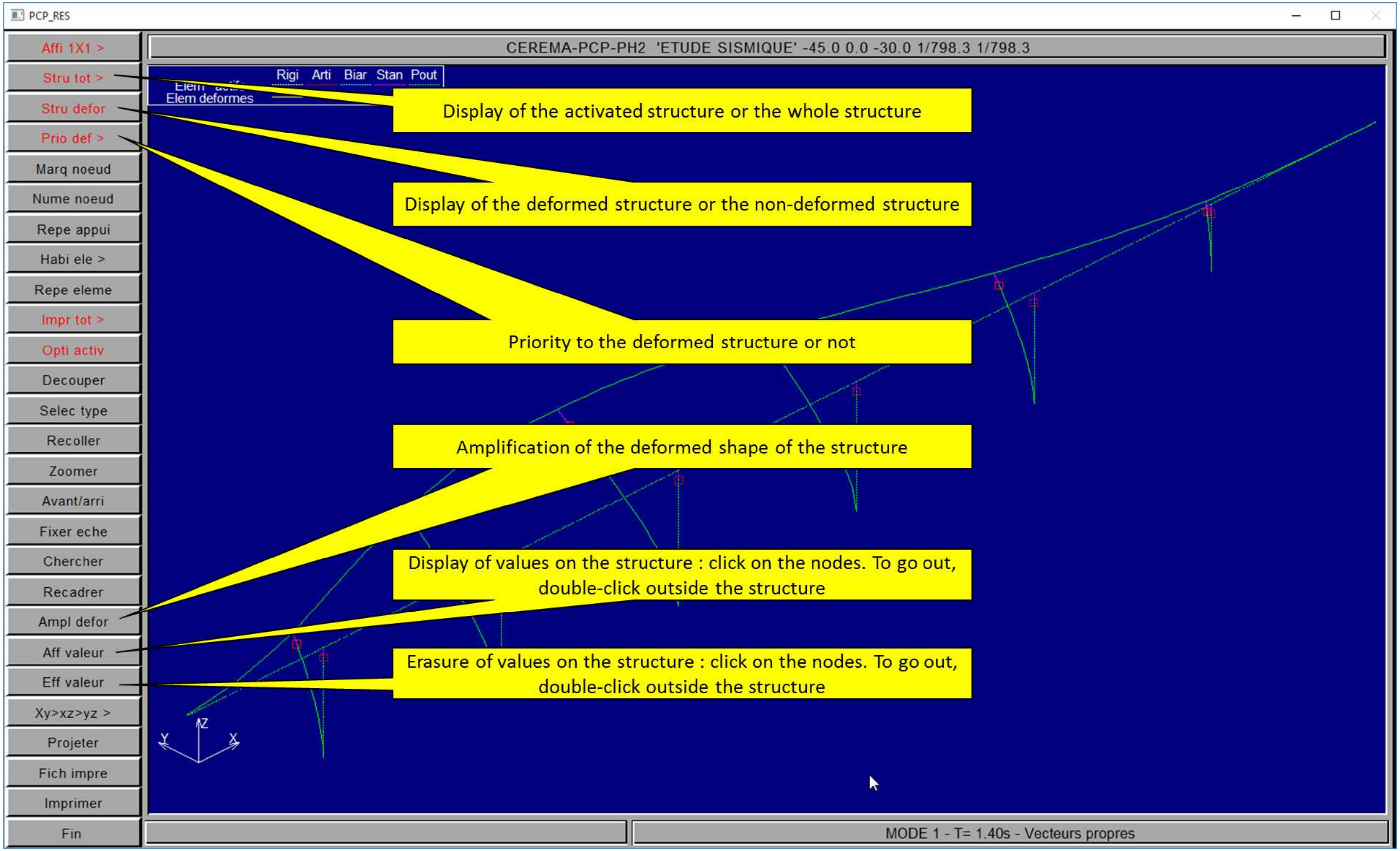


Display in graph form. Abscissa is generally geometrical, but for evolutions and vibrations, absissa is temporal. In that case, the study points are needed to visualize their variations.

Display of displacements or other effects on structure. In the cases of evolutions and vibrations, only displacements can be visualized because of the dynamical display. For the MODES, the display can be static or dynamical depending on the number of cycles asked : 0 for a static view. In all the other cases, a static view is displayed.

Visualization of results in table form : for one effect and one study.

Results : display of displacements of the structure



CEREMA-PCP-PH2 'ETUDE SISMIQUE' -45.0 0.0 -30.0 1/798.3 1/798.3

MODE 1 - T= 1.40s - Vecteurs propres

Display of the activated structure or the whole structure

Display of the deformed structure or the non-deformed structure

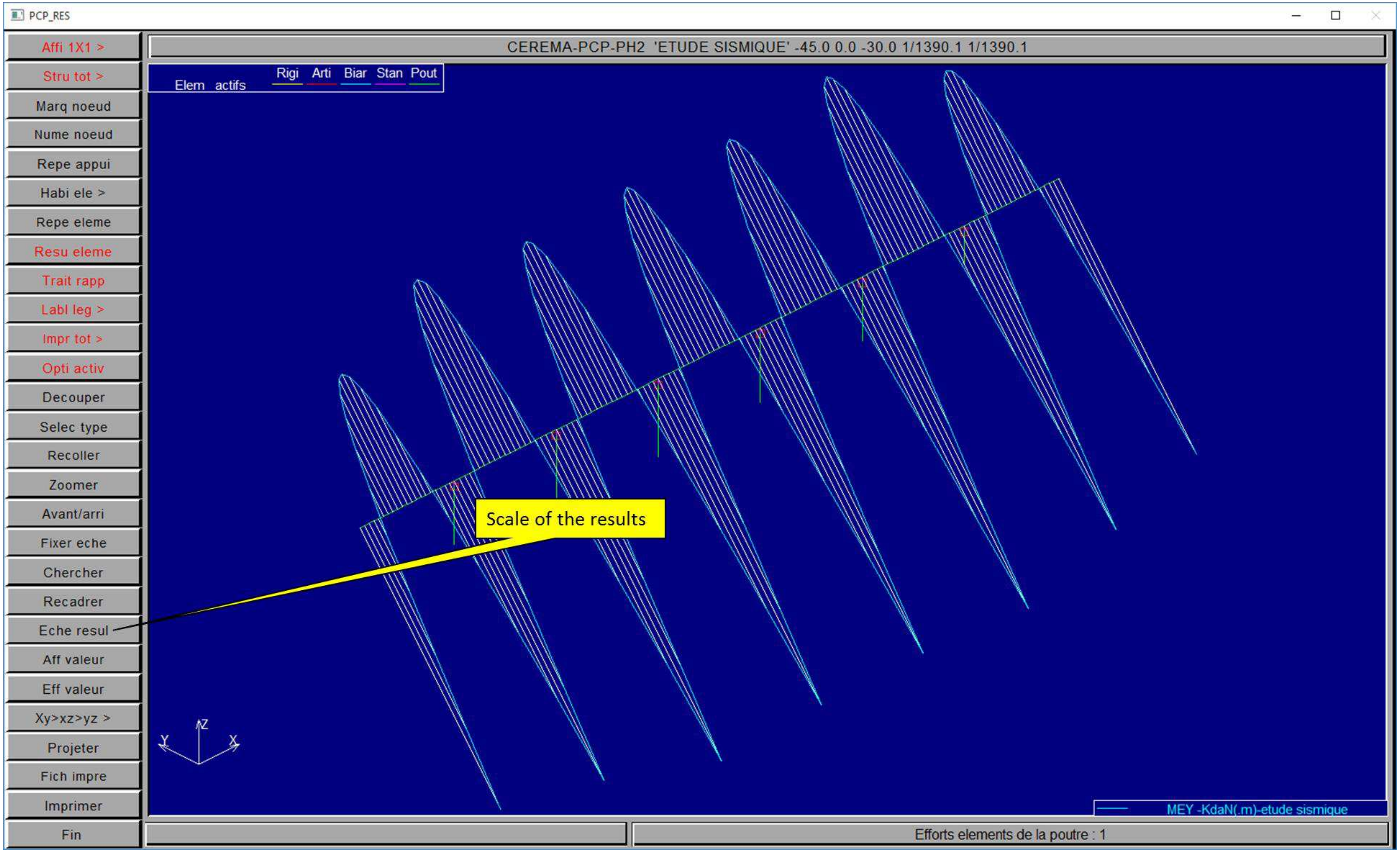
Priority to the deformed structure or not

Amplification of the deformed shape of the structure

Display of values on the structure : click on the nodes. To go out, double-click outside the structure

Erasure of values on the structure : click on the nodes. To go out, double-click outside the structure

Results : display of effects on the structure

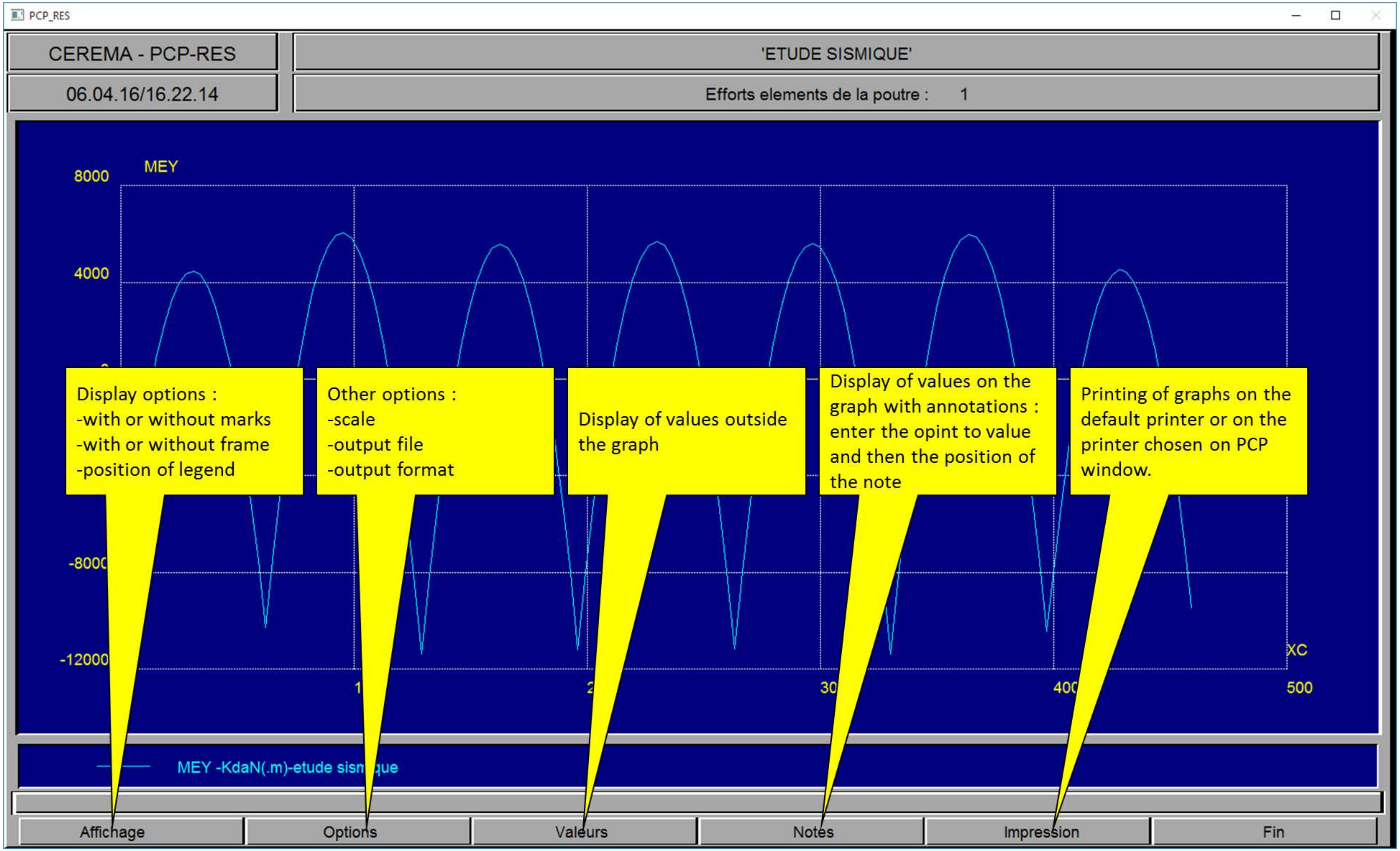


CEREMA-PCP-PH2 'ETUDE SISMIQUE' -45.0 0.0 -30.0 1/1390.1 1/1390.1

Efforts elements de la poutre : 1

Scale of the results

Results : display of effects as a graph



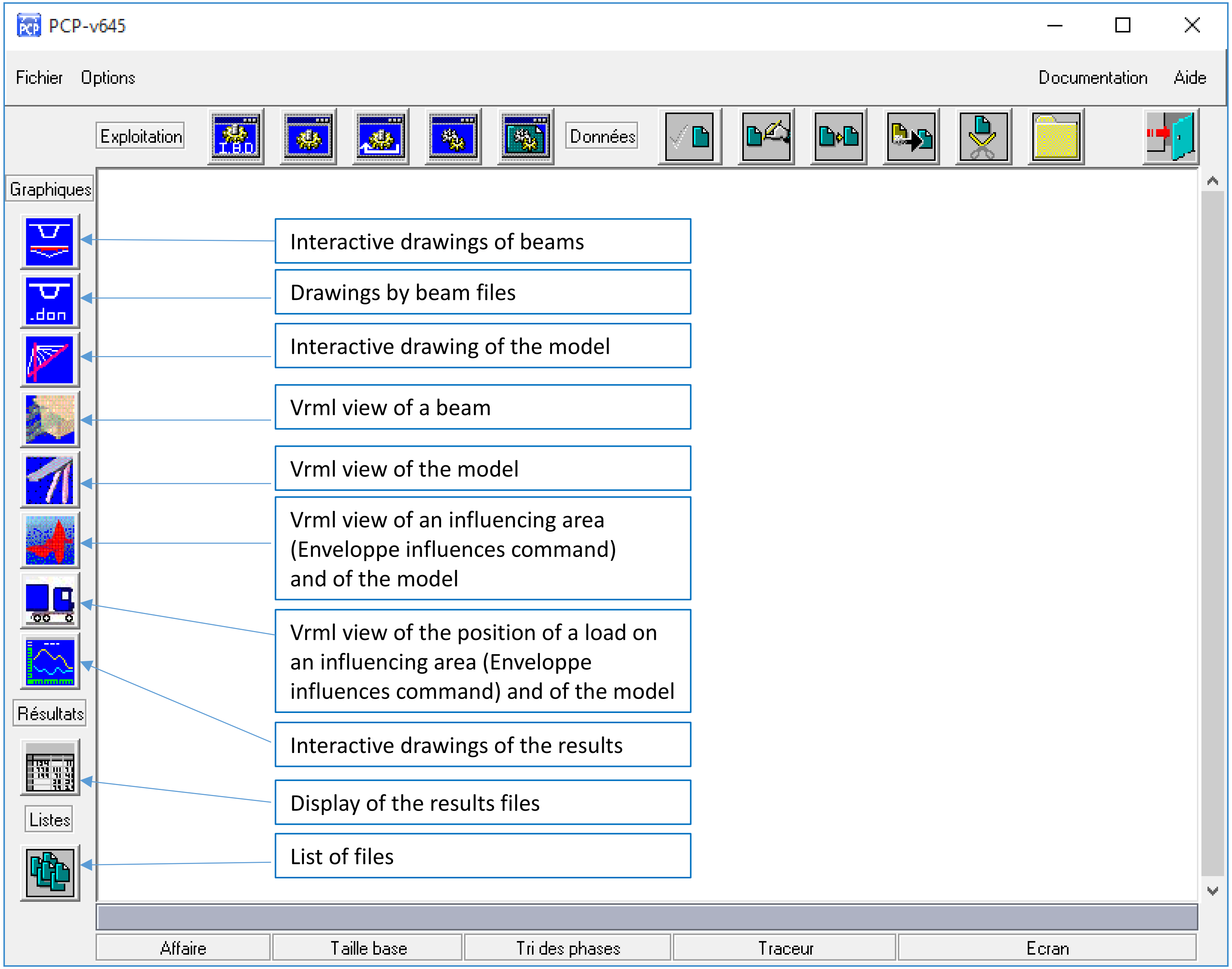
Display options :
-with or without marks
-with or without frame
-position of legend

Other options :
-scale
-output file
-output format

Display of values outside the graph

Display of values on the graph with annotations : enter the opint to value and then the position of the note

Printing of graphs on the default printer or on the printer chosen on PCP window.



Interactive drawings of beams

Drawings by beam files

Interactive drawing of the model

Vrml view of a beam

Vrml view of the model

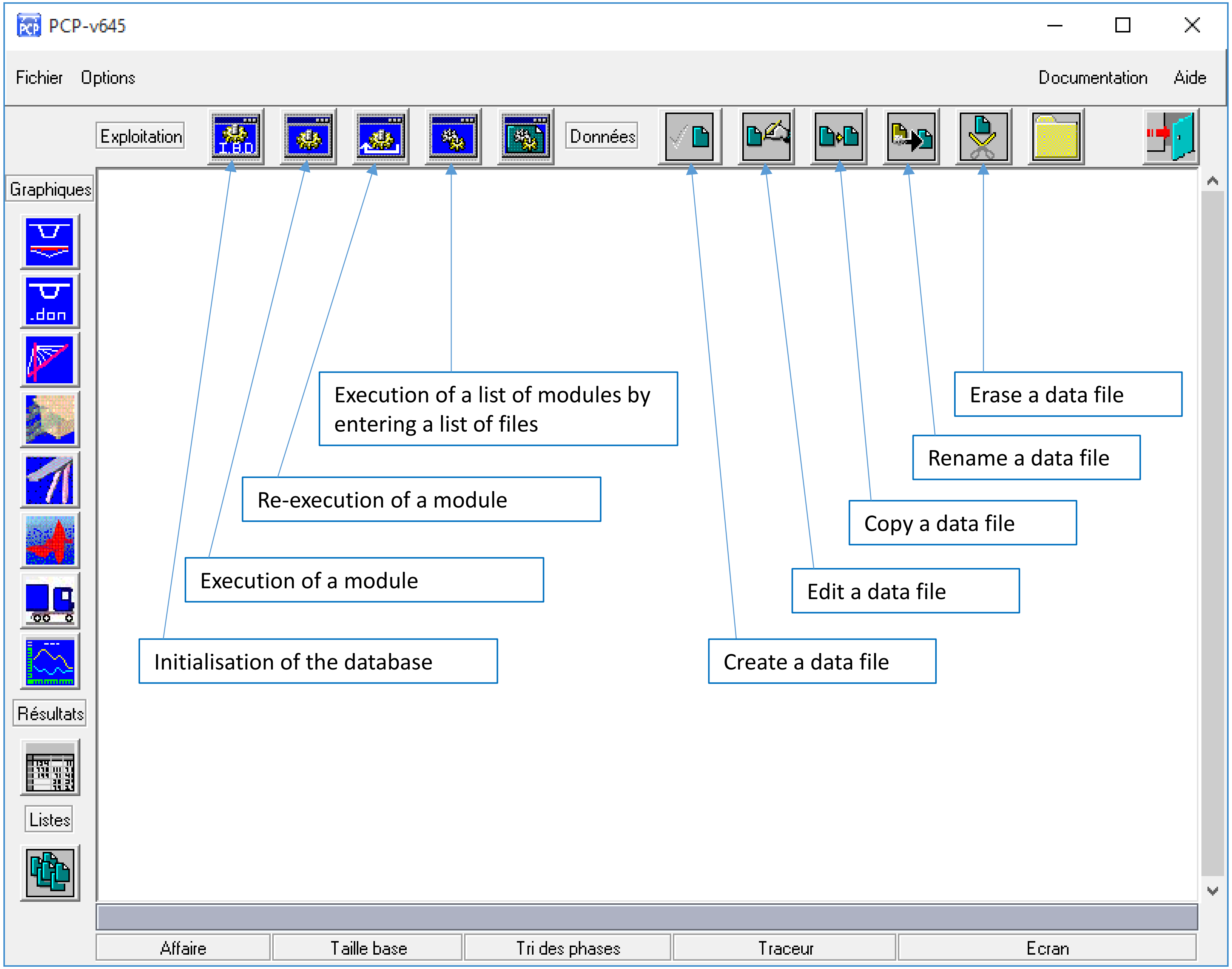
Vrml view of an influencing area (Enveloppe influences command) and of the model

Vrml view of the position of a load on an influencing area (Enveloppe influences command) and of the model

Interactive drawings of the results

Display of the results files

List of files



Initialisation of the database

Execution of a module

Re-execution of a module

Execution of a list of modules by entering a list of files

Create a data file

Edit a data file

Copy a data file

Rename a data file

Erase a data file

Introduction

0.1 - GENERAL

The PCP software (Phased Bridge Construction) is a set of modeling and analysis programs for THREE DIMENSIONAL BAR-based structures, possibly part PRESTRESSED, particularly suited to the study and verification of ENGINEERING STRUCTURES.

The distribution, intensity, and the history of occurrence of extreme loading in such works often depend closely on:

- their mode of implementation (CONSTRUCTION PHASING)
- the linear, or nonlinear rheological BEHAVIOUR, of certain of their constituent MATERIALS;
- the effects of TRAFFIC loads (road, rail, etc.), the WIND, or EARTHQUAKES.

Despite its increasingly generalist nature, the PCP software preferentially exploits this niche use, thanks to its ability to reproduce by simulation most of the methods of construction and operating load systems, to meet various regulations, and its potential for dynamic analysis.

From the perspective of its constitution and its use, the PCP software is characterized by:

- the direct acquisition of geometrical formwork and cabling data for prestressed beams, and their semi-automated insertion in modeling;
- the ability to graphically verify the majority of geometric data;
- its modularity, which allows the introduction and progressive validation of data;
- a description of the data using a free format command language, that is explicit for the structural engineer, using certain programming techniques;
- dynamic memory management (RAM and mass), which ensures almost no limits in many cases;
- the ability to store, edit and/or graph, on request, most of the essential results, independently, or combined with modeling.

Through its programming mode and the multi-peripheral standardized graphics core it uses, PCP software is easily transportable, from one operating system to another.

0.2 - THEORIES AND METHODS USED

The main methods of calculation are based on the Navier Bresse beam theory, therefore:

- the plane sections conservation law is applied;
- the straight sections are assumed to be transversely undeformable and constructed in one stage, using a homogeneous material;
- centers of gravity and of torsion of straight sections may be different; loading however is provided at their centers of gravity;
- materials are assumed to be elastic and with linear rheological behavior; the possible variation of their characteristics over time gives rise to phenomena of adaptation by shrinkage, creep and relaxation; in this case, the principle of superposition of constraints is permitted;
- under these assumptions, the material behavior laws are arbitrary; in practice, they are taken from various regulations;
- material nonlinearity is considered as an option, with the parameters of the elastoplastic constitutive laws of the materials being provided in addition;
- when the default assumption of "small" displacements is not retained or is inapplicable, geometric nonlinearity can be taken into account, by considering large rotations and large displacements, according to the method of semi-inverse convected co-rotational analysis;
- buckling can be analyzed using the linear method, which allows extraction of the various modes, or by the incremental method, which takes into account all the nonlinearities;
- the theory of thin-walled profiles is used to calculate the tangential stresses (still in linear elastic mode), but the effect of bi-moments is omitted.

The displacement method used for structural analysis is well suited to the study of structures with complex and evolving static patterns, and to effective computer resolution of the systems of equations derived from modeling.

Dynamic calculations are based on the modal analysis of structures. The effects of turbulent winds are deducted from their statistical characteristics and from this modal decomposition, by applying a spectral method. Those for earthquakes are based on modal superposition of responses from regulatory or arbitrary spectra.

Cooley's Law is used to calculate the instantaneous tension loss in the prestressing cables, but local effects, particularly the diffusion of prestressing forces, are not taken into account.

0.3 - STRUCTURAL MODELING

The possible structures for processing may include one or more beams with a straight or curved mean fiber, called "spatial beams" and/or straight prismatic elements connecting these beams, or constituting an autonomous network, to which point, perfect or elastic internal articulations and perfect or elastic supports may be added.

Spatial beams

Their sections, whose characteristics can vary longitudinally, are treated as thin-walled profiles, or regarded as thick.

They are discretized, according to a cut-up procedure fixed in the data, into straight prismatic elements, whose end section centers of gravity constitute the modeling nodes (and possible junction with other elements), and whose mechanical characteristics are calculated .

Displacements of the nodes from their mean fibers are calculated in the global reference frame.

The corresponding forces, calculated in the local reference frame of their elements and projected onto the reference frame of their straight sections, will be the basis for calculating the normal and tangential stresses at various specified points.

When a spatial beam is prestressed, the instantaneous tension losses incurred by its cables due to friction, tensioning methods and anchor retreat are calculated and deducted from the equivalent initial loads to be applied to it.

Other major benefits of the inclusion of part of a structure as a spatial beam are as follows:

- its formwork and cabling can be drawn independently;
- the formwork can be described in a suitable reference frame, and repositioned in the global reference frame;
- it can be duplicated and repositioned to generate other spatial beams;
- its formwork reference fiber can be used to position certain of the general mechanical model nodes, situated outside its mean fiber;
- it can be drawn as a substructure of the general mechanical model;
- it can be selected as a component of a push-launch structure;
- it can be loaded (direct or thermal), the results edited, or special calculations performed (envelope effects, general processing);
- corrections to tangential stresses that develop there, due to the Résal effect, can be calculated automatically;
- the geometry of its sections allows nonlinear materials calculations;
- its formwork reference fiber can participate in the definition of a mobile load carrier, its mean fiber being the carrier structure;
- aerodynamic masses or characteristics can be globally assigned to it, for dynamic studies;
- admissible tangential stresses can be determined, at certain points of calculation of its sections, according to various criteria;
- its transverse reinforcing steel, required to meet various regulatory criteria, can be determined on request;
- its sections can be extracted, with their cables and loads applied, and transferred to the CDS software for limit state calculations.

Straight prismatic elements

These "non-beam" elements are introduced directly, in the form of "simple" data with node coordinates (provided in the global reference frame or in the reference frames defining certain beam sections), incidence and properties of elements.

Current items are called "standard" when embedded in their incident nodes or "bi-articulated" (with possible "chain-like" behavior) when the three rotations of their incident nodes are free.

Certain non-current elements can be "infinitely stiff", declared explicitly, or implicitly as eccentricities of current elements.

The displacements of their incident nodes are calculated in the global reference frame.

The corresponding forces are calculated in their local reference frames.

Modelizable structures

The spatial beams and/or the non-beam elements, assembled in any way, enable structures such as the following to be processed:

- caisson bridges;
- ribbed slab bridges;
- cable-stayed bridges;
- arch bridges, upper, middle, or lower deck;
- truss bridges;
- suspension bridges;
- composite bridges;
- rigid frame bridges, etc.

Or any combination of the above.

0.4 – CONSTRUCTION PHASES

Rigorous simulation of the construction process (in successive phases) of a structure to determine its exact state of deformation and loading, before applying operating loads.

At each phase of construction, the structure corresponding to the previous static scheme (active or inactive elements, links in place) may be modified by adding or removing elements or links.

Potential additional applied loads are mainly the result of the effects of:

- the dead weight of the elements;
- prestressing;
- external loads.

And the following results can be edited (for the active part of the general mechanical model):

- the effects produced by the current construction phase, on the deformations and loadings;
- the resulting state of deformation and loading;
- the extreme values of certain loadings, since the start of construction;
- tension in the prestressing cables.

If a history of the construction phases is provided, the phenomena of creep and shrinkage of concrete, relaxation of prestressing steels, and their interactions, are taken into account precisely.

Otherwise, a regulatory standard method is proposed to evaluate delayed prestressing losses.

The viscoelastic and nonlinear behaviors of the materials are compatible with each other and with the geometric nonlinearity of the structures; these phenomena can be combined without restriction in the calculations.

In a linear calculation of buckling, a critical factor applicable to all loads (including gravity) is calculated for each mode; the corresponding deformation can be introduced as an "initial" deformation.

When calculating nonlinear buckling (only able to take into account material nonlinearity), the rupture threshold applicable to the current case load is determined by an incremental method.

Because of its generality, PCP software can simulate implementation techniques as diverse as construction using:

- hangers, incremental advancement;
- balanced cantilever;
- cabling, temporary and/or permanent;
- launching, etc.

Or any combination of the above.

0.5 – OPERATING LOADS

These loads, fixed or mobile, apply to a structure whose static diagram is frozen during the simulation of the construction, most often until the end.

At certain points of the selected structure, elastic envelope effects resulting from the application of operating loads, meeting certain French or European traffic overload regulations, or redefined, are calculated and stored.

The predefined regulatory loads repertoire can be enriched by the addition of arbitrary traffic loads (rail or otherwise), without real limitation to this possibility.

0.6 - DYNAMIC STUDY

The structure to which the dynamic study is applied is also frozen during the simulation of the construction, with a distribution of masses resulting from its definition that can be supplemented.

Some of its natural modes of vibration, obtained by modal analysis, are selected and may be stored.

The damping coefficients and the aerodynamic characteristics of the elements being introduced and the response of the structure to a defined turbulent wind can be obtained by spectral analysis and stored.

Seismic study of the structure is also feasible, using the results of modal analysis and regulatory or arbitrary response spectra to be taken into account.

0.7 – PROCESSING OF RESULTS

The main compatible effects, stored during simulation of construction (structural states, load effects or buckling modes), following the application of operating loads (envelope effects) or the dynamic study (vibration eigenmodes, modal responses) can be reread, edited, completed by directly acquired effects, combined, weighted and/or enveloped.

The results of this processing can be edited, exported, stored and/or reprocessed in the same way.

For certain spatial beam sections, limit state calculations will be made possible, and minimum quantities of transverse reinforcing steel can be obtained.

All stored results can be visualized, in the form of graphs or superimposed on the general mechanical model drawings.

0.8 - GENERAL ORGANIZATION

All PCP software modules communicate via a central file called the "database" that represents a structure or a "contract". In the diagram below, only the utility modules (management of the database) are not represented.

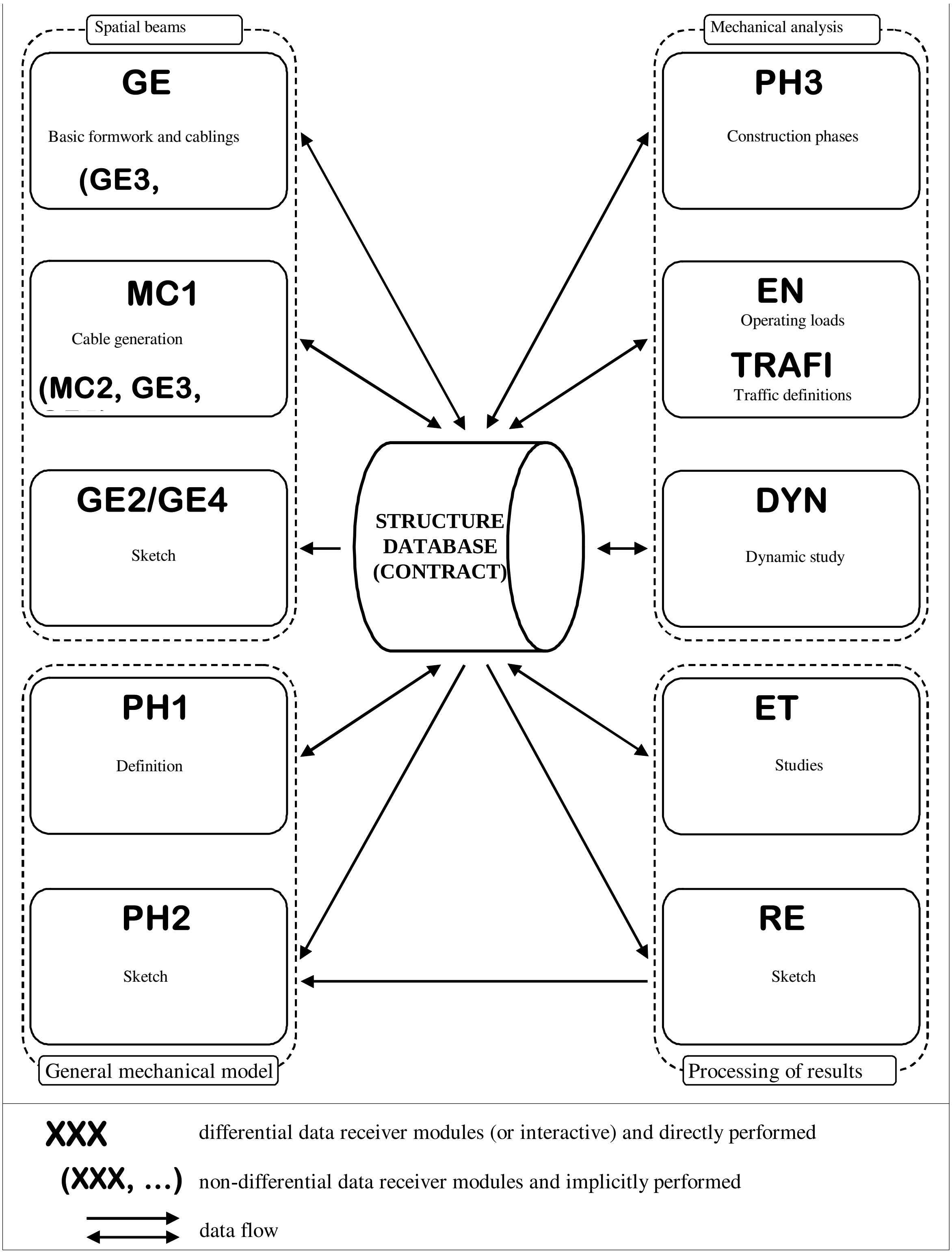


Figure 0.1 - PCP software, modular structure

0.9 - GRAPHIC OUTPUT EXAMPLES

This section contains a representative selection of the main drawings that the GE2, GE4, PH2 and RES modules can produce.

They can be printed on media of different formats, of which landscape A4 is the smallest and most widespread.

For aesthetic and space-saving reasons, these designs have undergone a shift and a reduction, without changing certain parameter values applying to them in absolute mode (text sizes and presentation details).

We will thus observe some distortions between the designs presented and those obtained under real operating conditions.

GE2 and GE4 modules

The selected spatial beam is the deck of a straight three-span bridge built by balanced cantilever method (the most common case), the cross sections are assumed to satisfy the theory of thin-walled profiles, and modeled as such.

PH2 module

The three-dimensional model chosen (suspension bridge that has undergone a replacement of its supporting cables and hangers) was reproduced in most forms of drawings that the PH2 module can produce, alone or in combination with the RES module to view the results of the mechanical analysis.

RES module

Graphs and area of influence represented apply to the same curved composite structure, which was modeled using four longitudinal spatial beams and connecting elements.

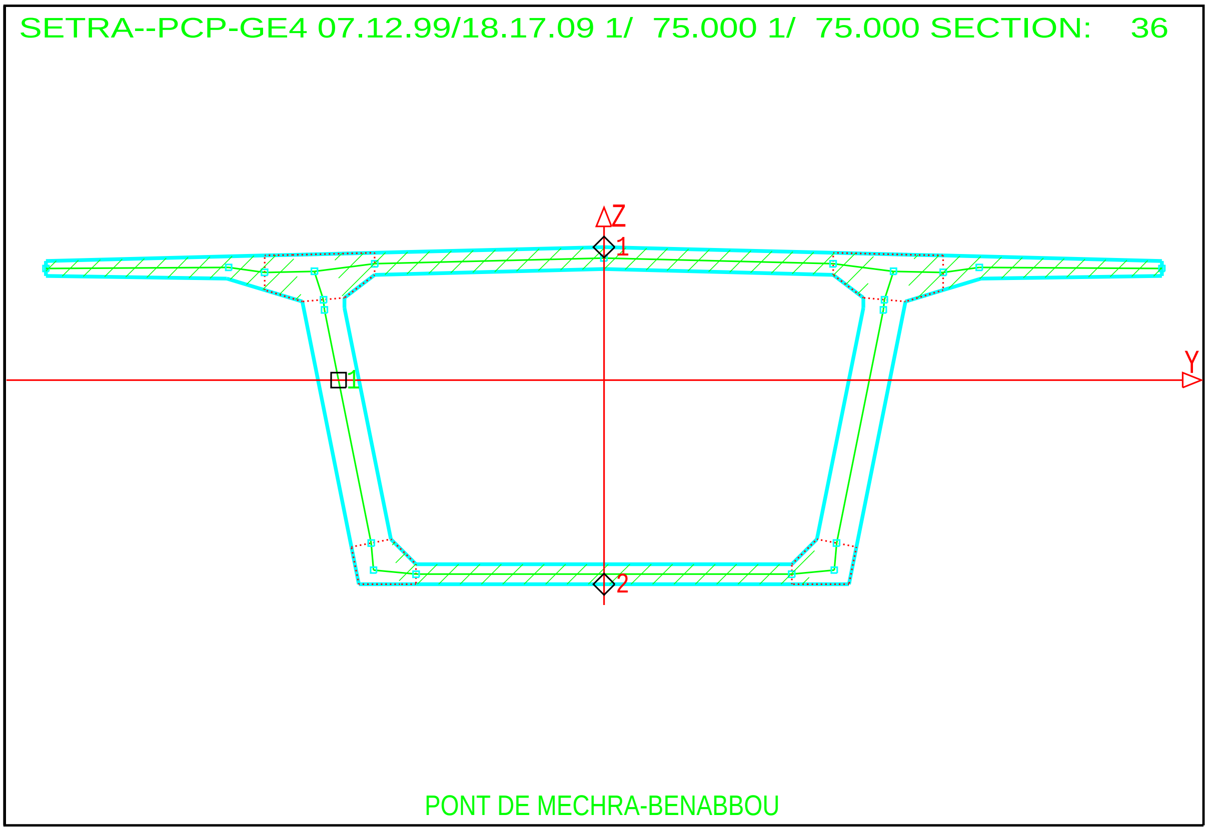


Figure 0.2 - Spatial Beam, cross section, formwork, thin walls

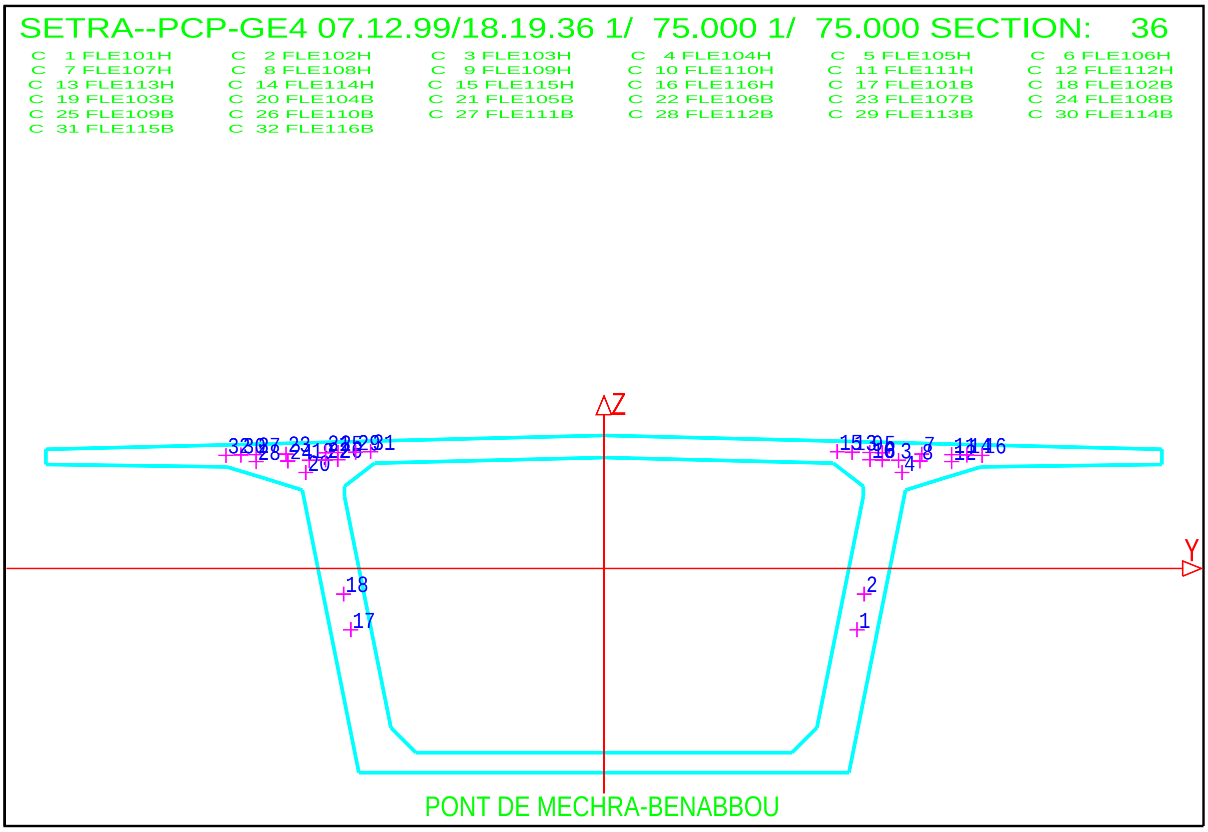


Figure 0.3 - Spatial Beam, cross section, formwork, cabling

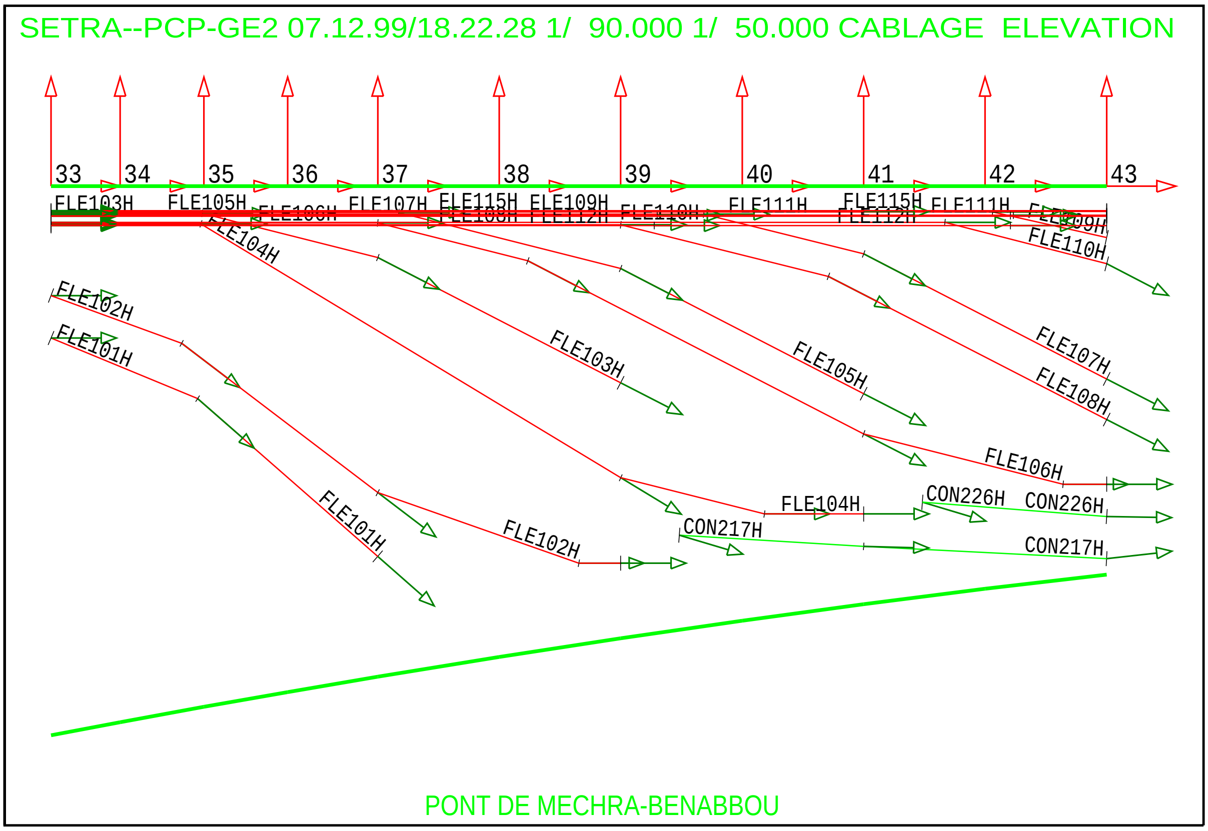


Figure 0.4 - Spatial beam section, longitudinal section 1

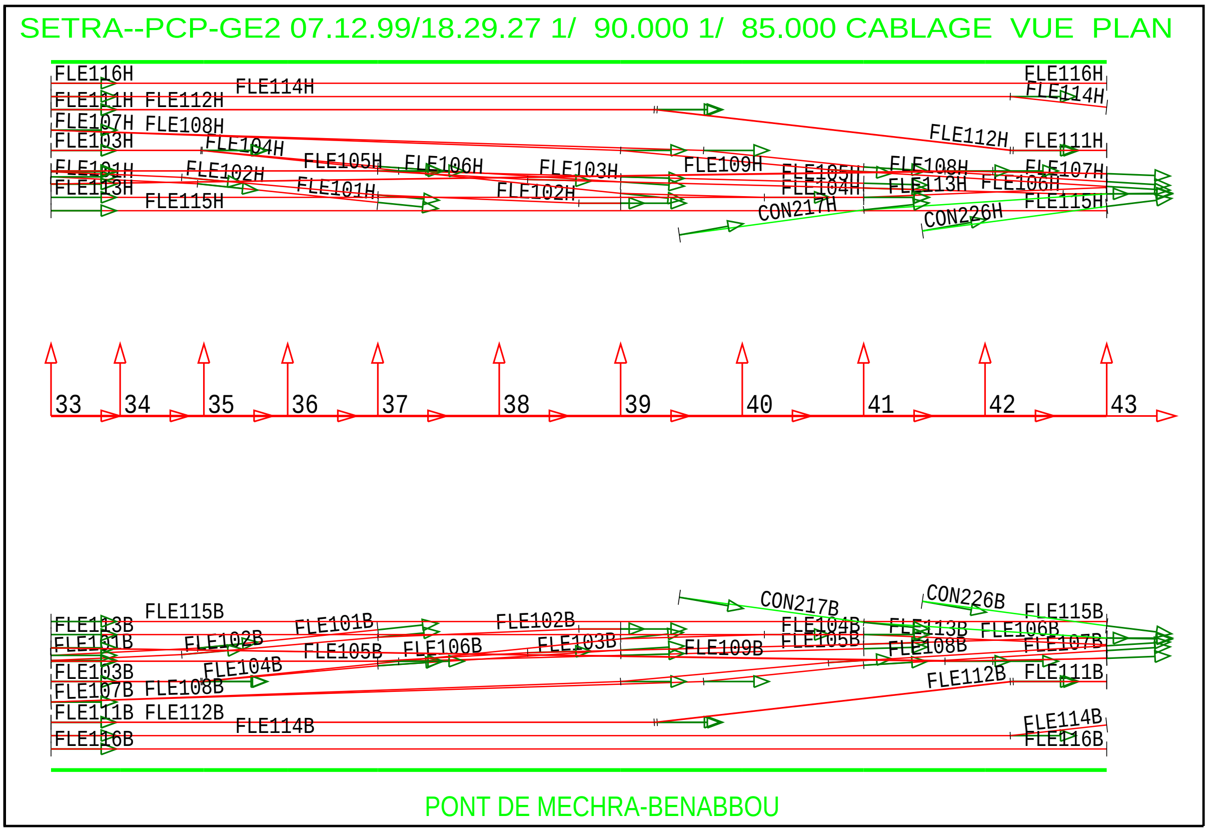


Figure 0.5 - Spatial beam section, longitudinal section 2

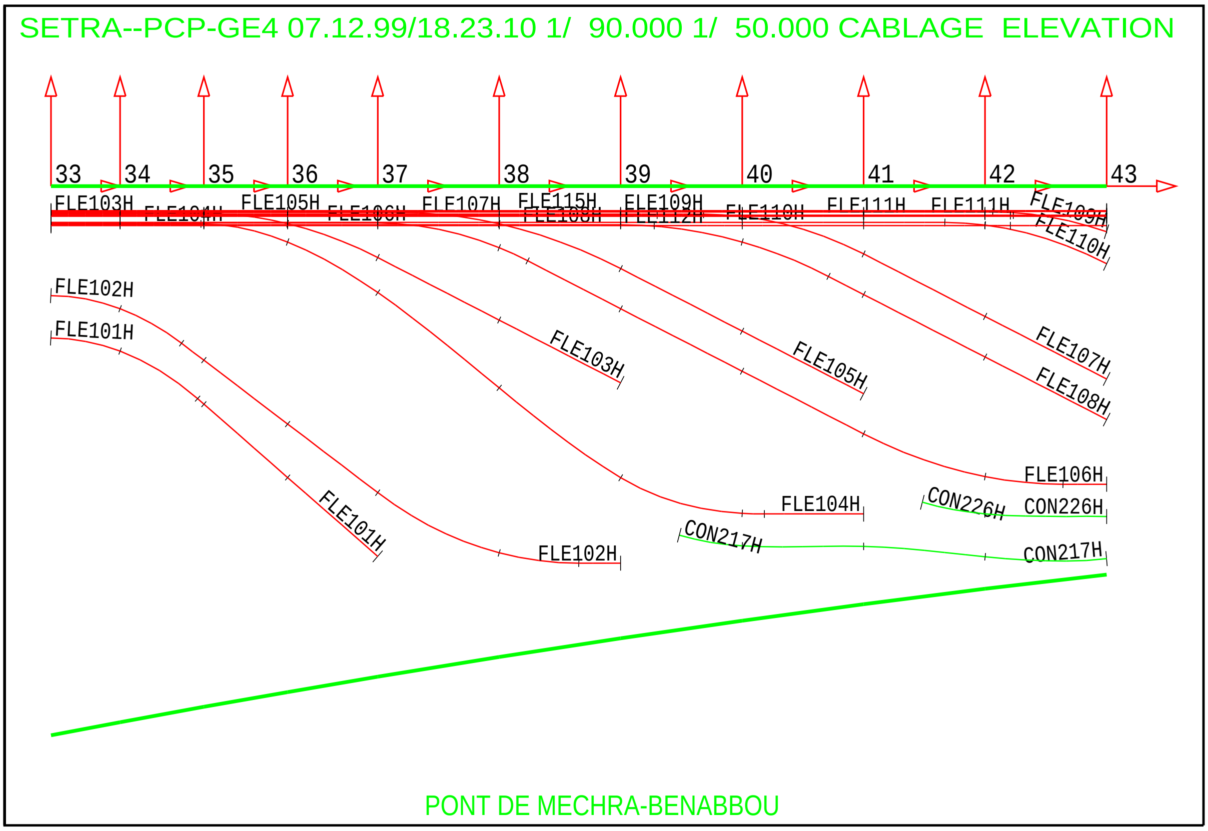


Figure 0.6 - Spatial beam section, longitudinal section 3

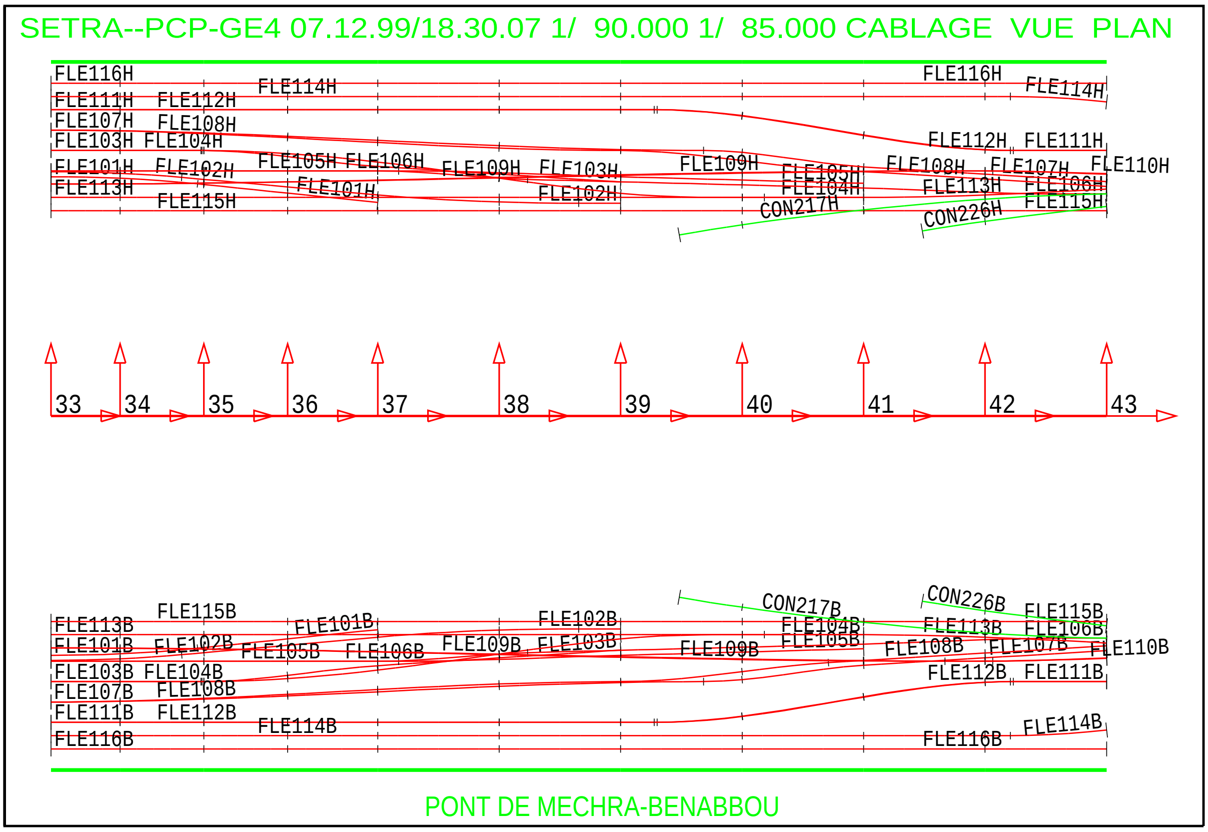


Figure 0.7 - Spatial beam section, longitudinal section 4

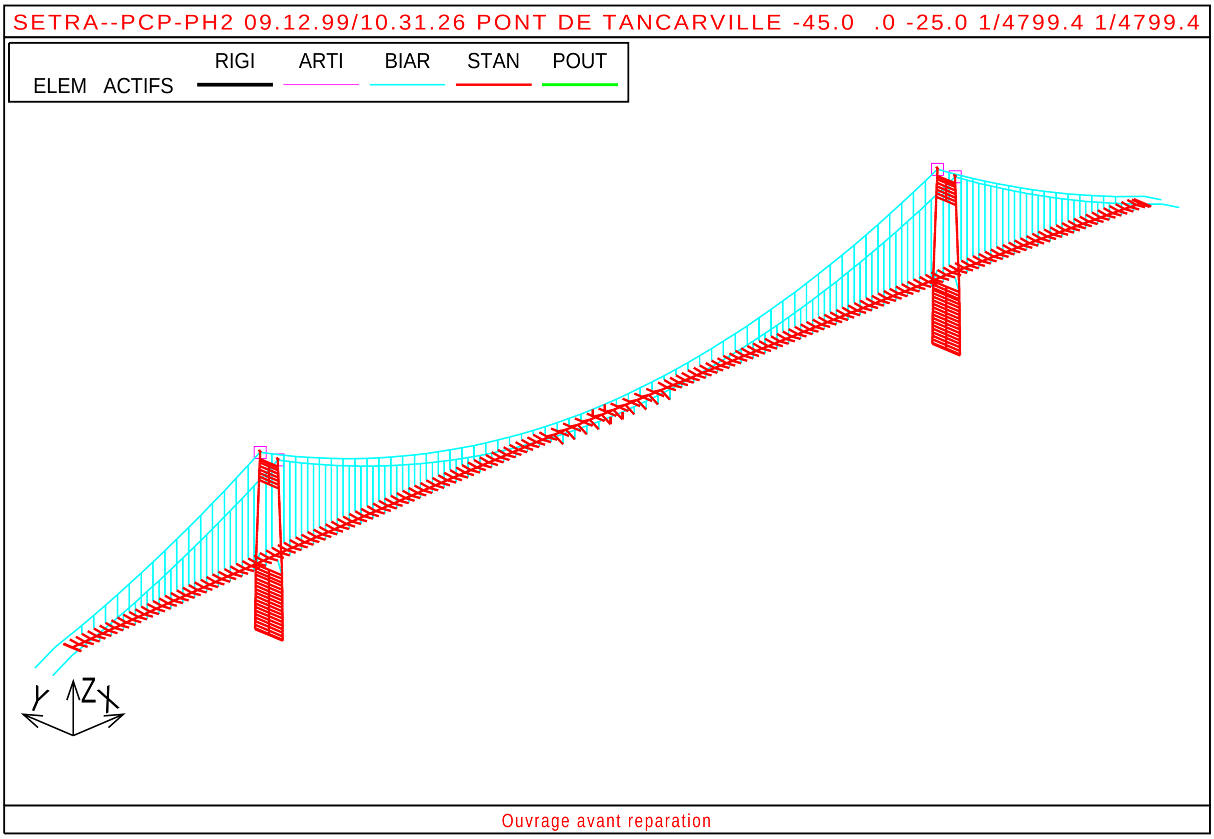


Figure 0.8 - General mechanical model, active part, simple perspective

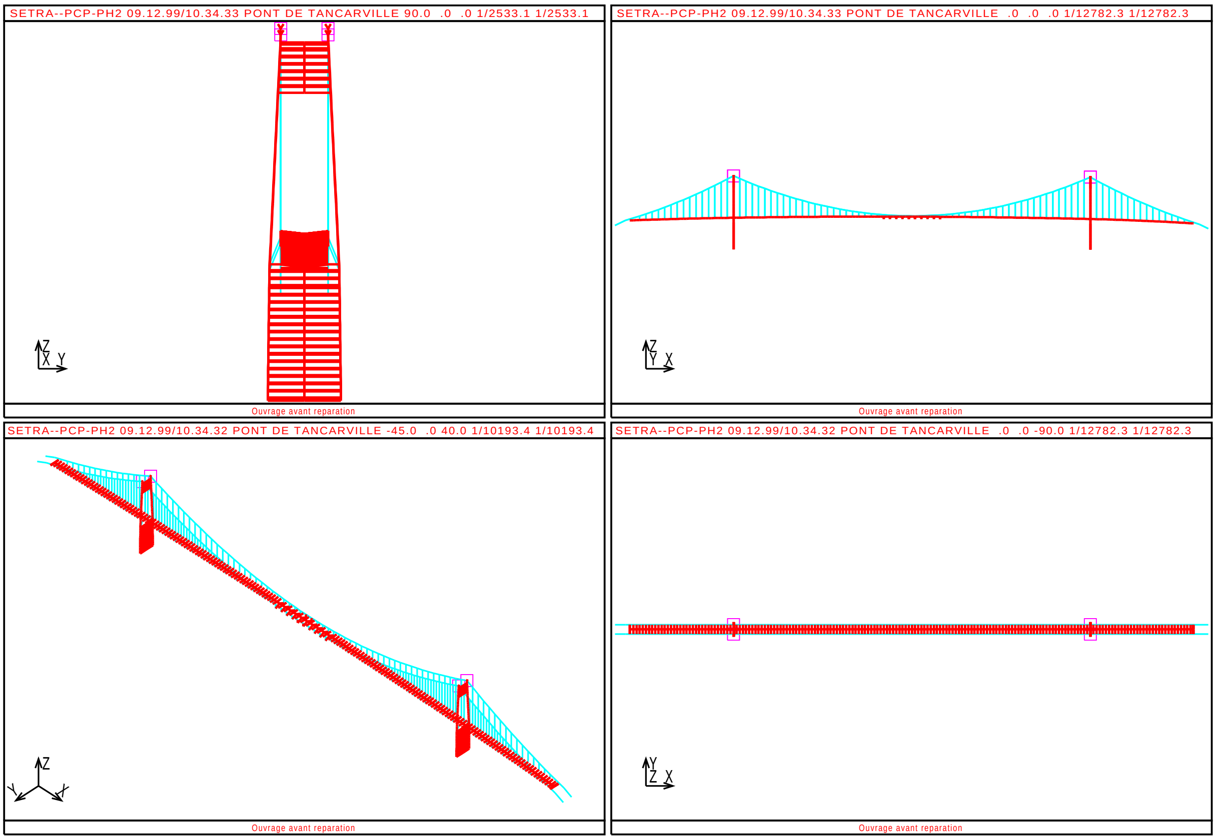


Figure 0.9 - General mechanical model, active part, four standard views

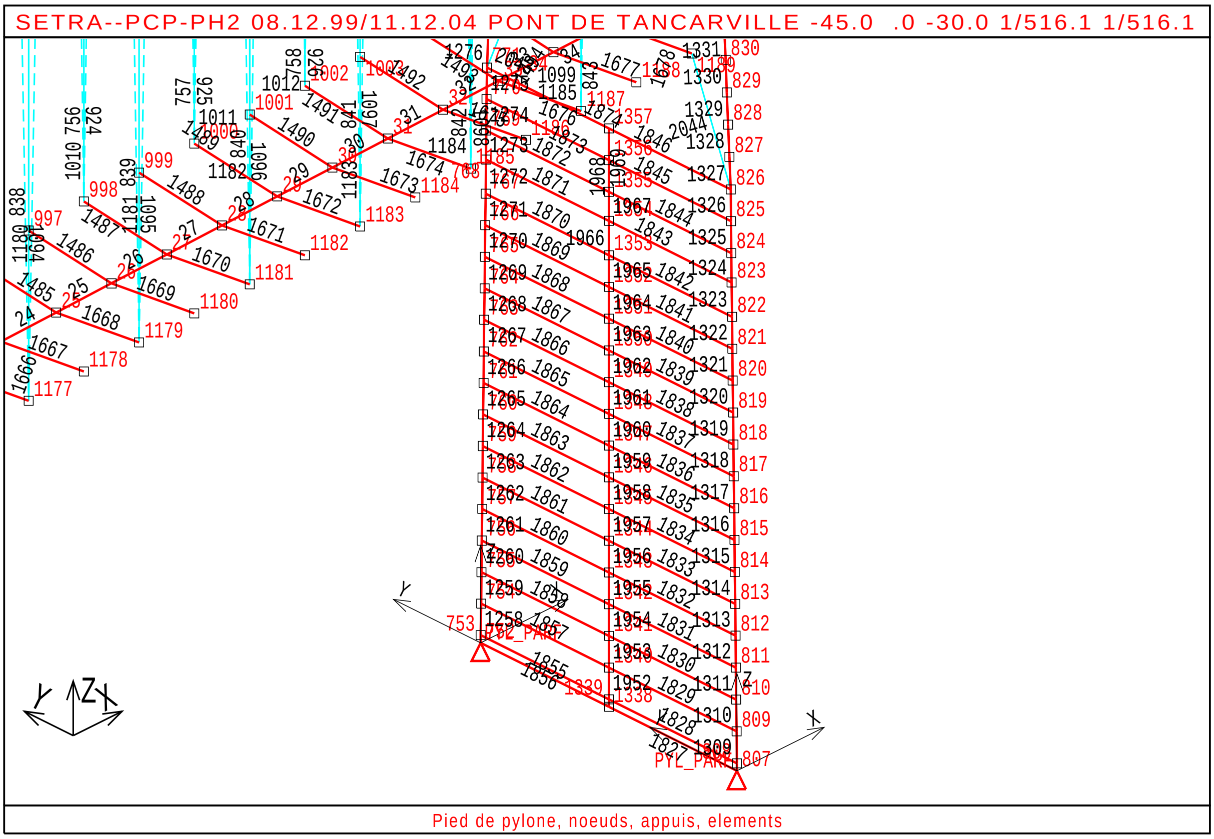


Figure 0.10 - General mechanical model, active and inactive parts, detail 1

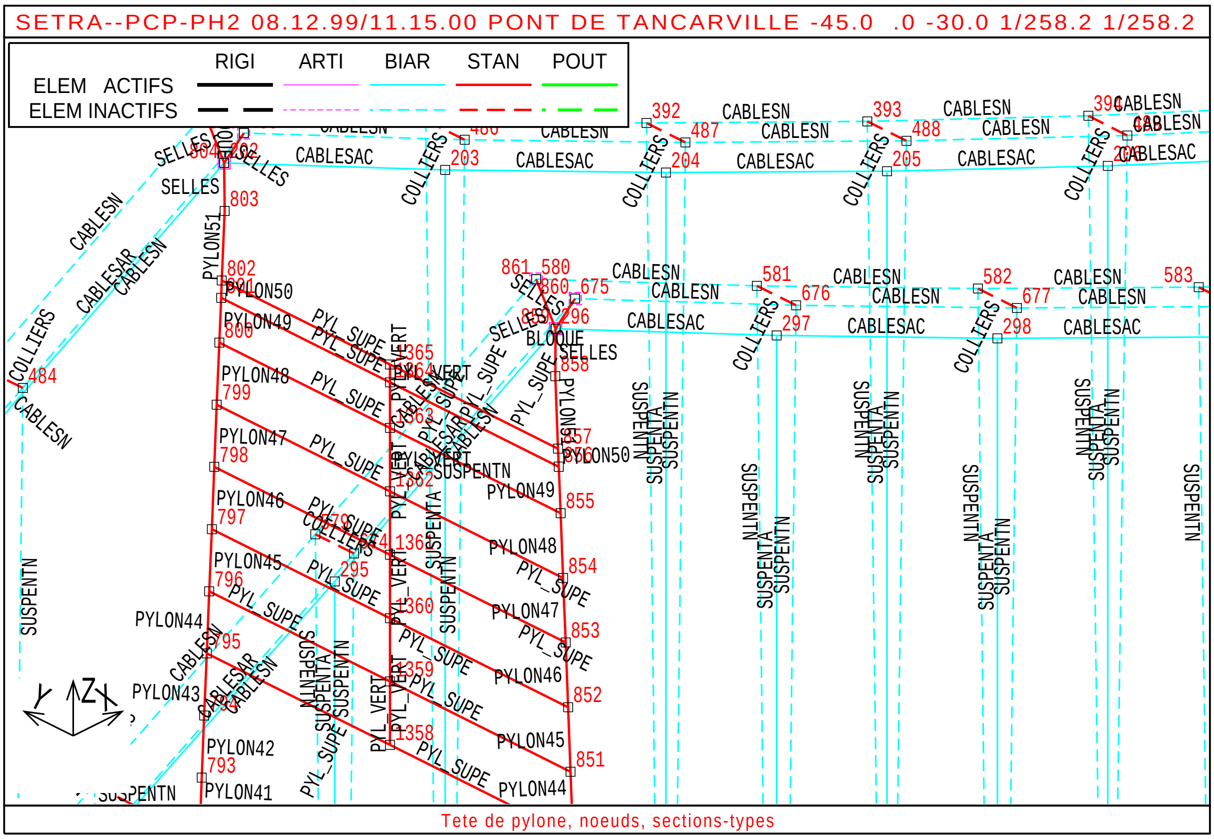


Figure 0.11 - General mechanical model, active and inactive parts, detail 2

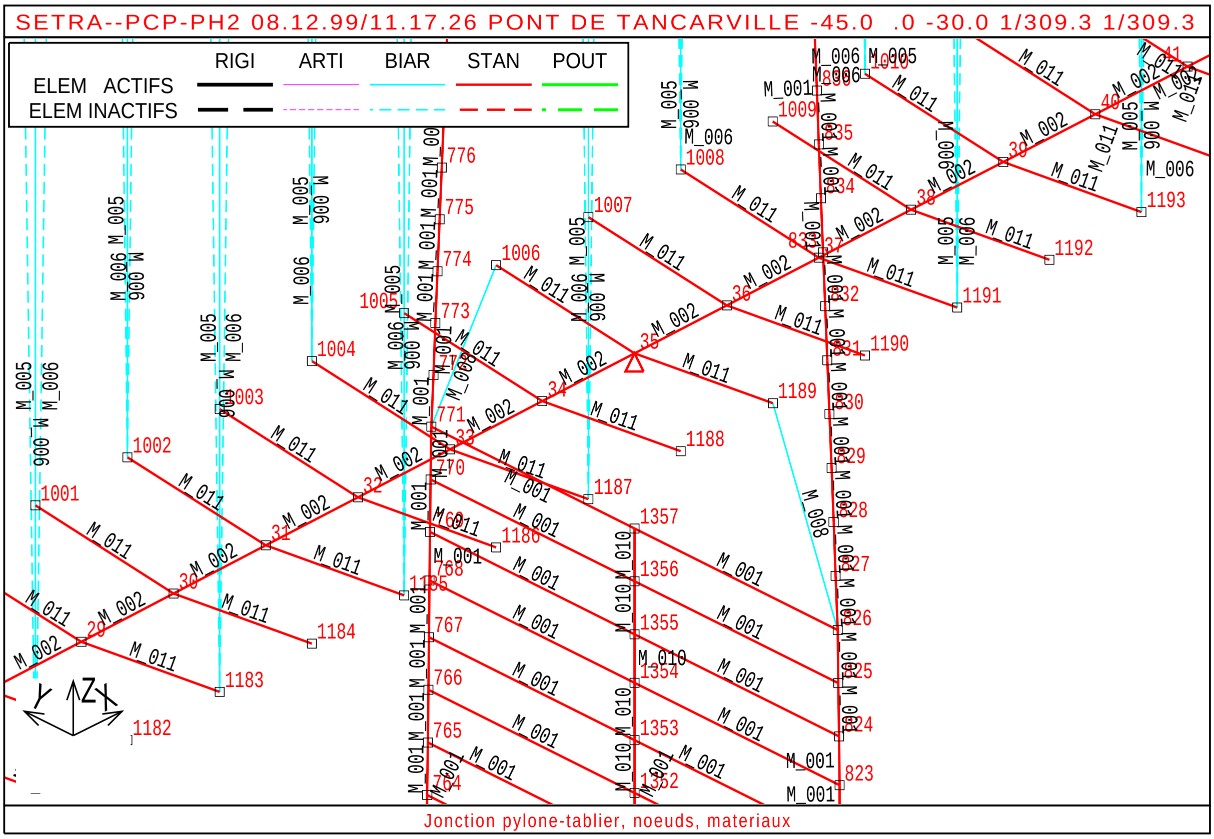


Figure 0.12 - General mechanical model, active and inactive parts, detail 3

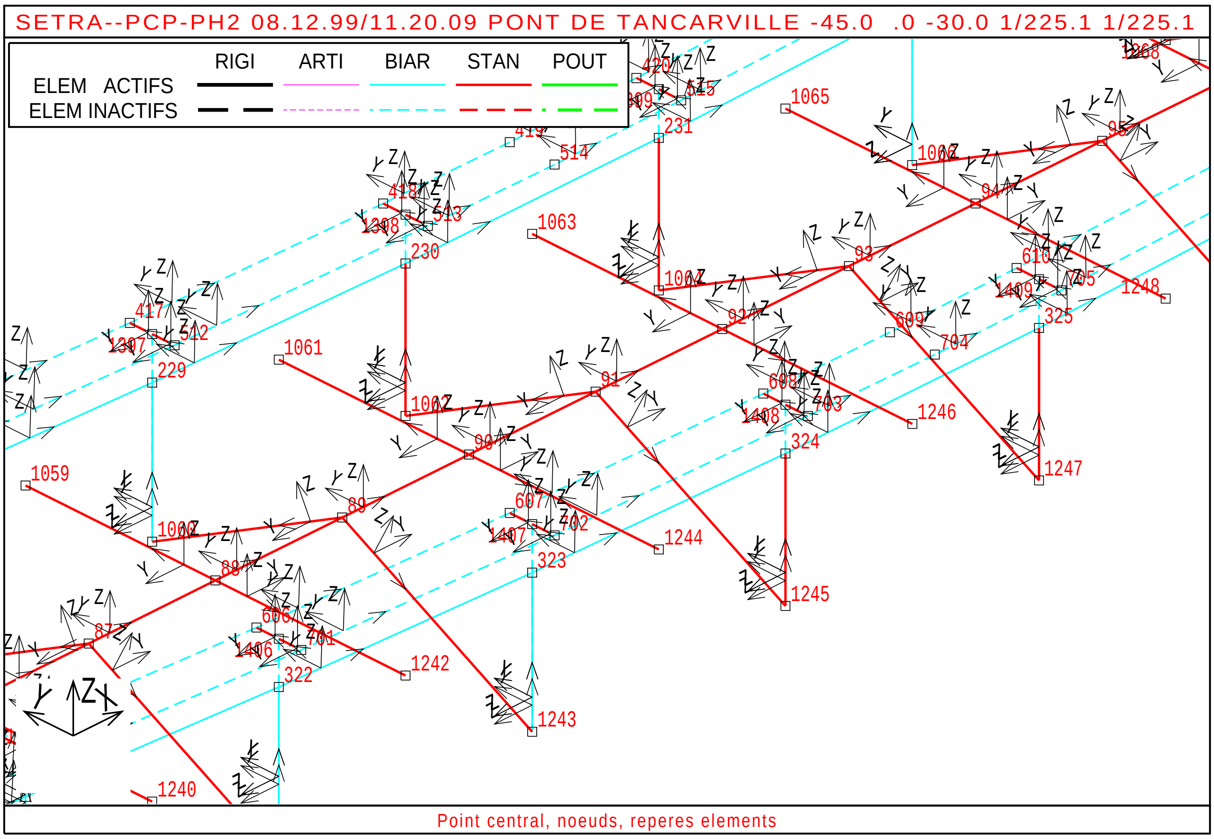


Figure 0.13 - General mechanical model, active and inactive parts, detail 4

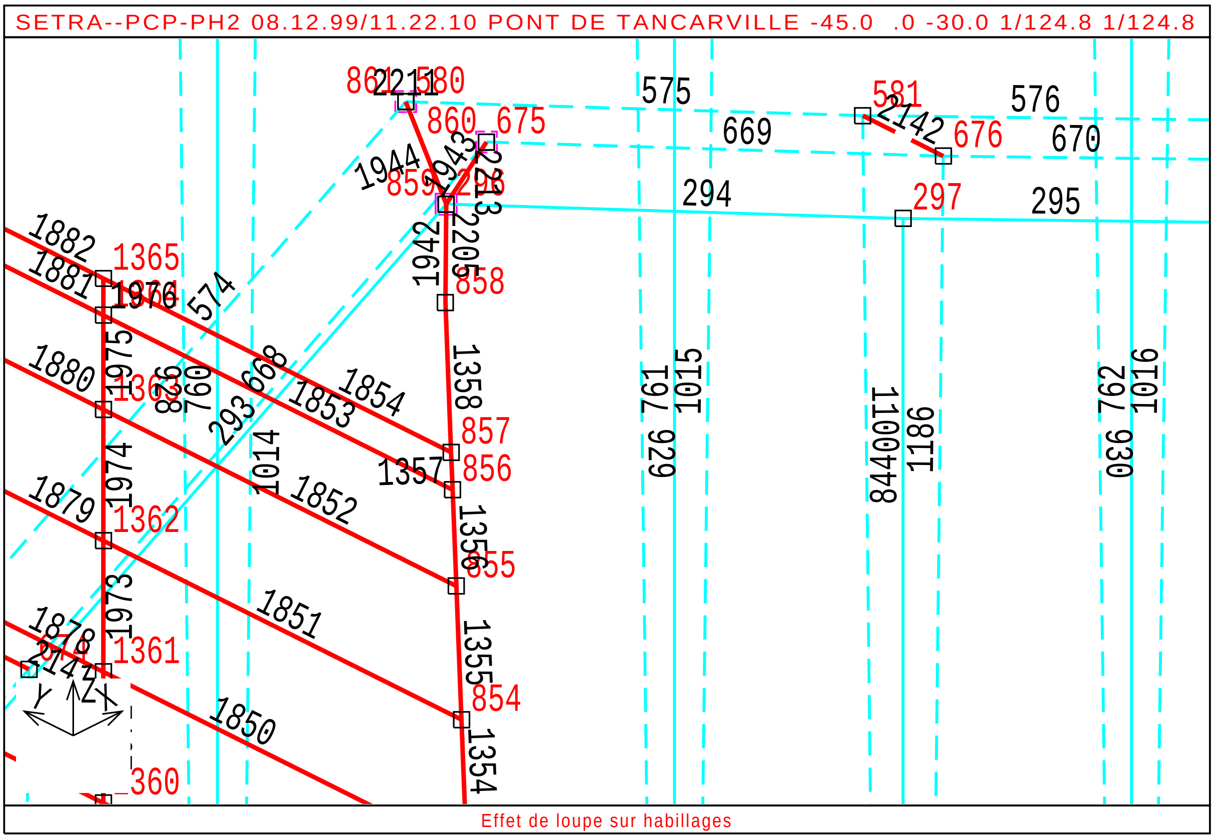


Figure 0.14 - General mechanical model, active and inactive parts, detail 5

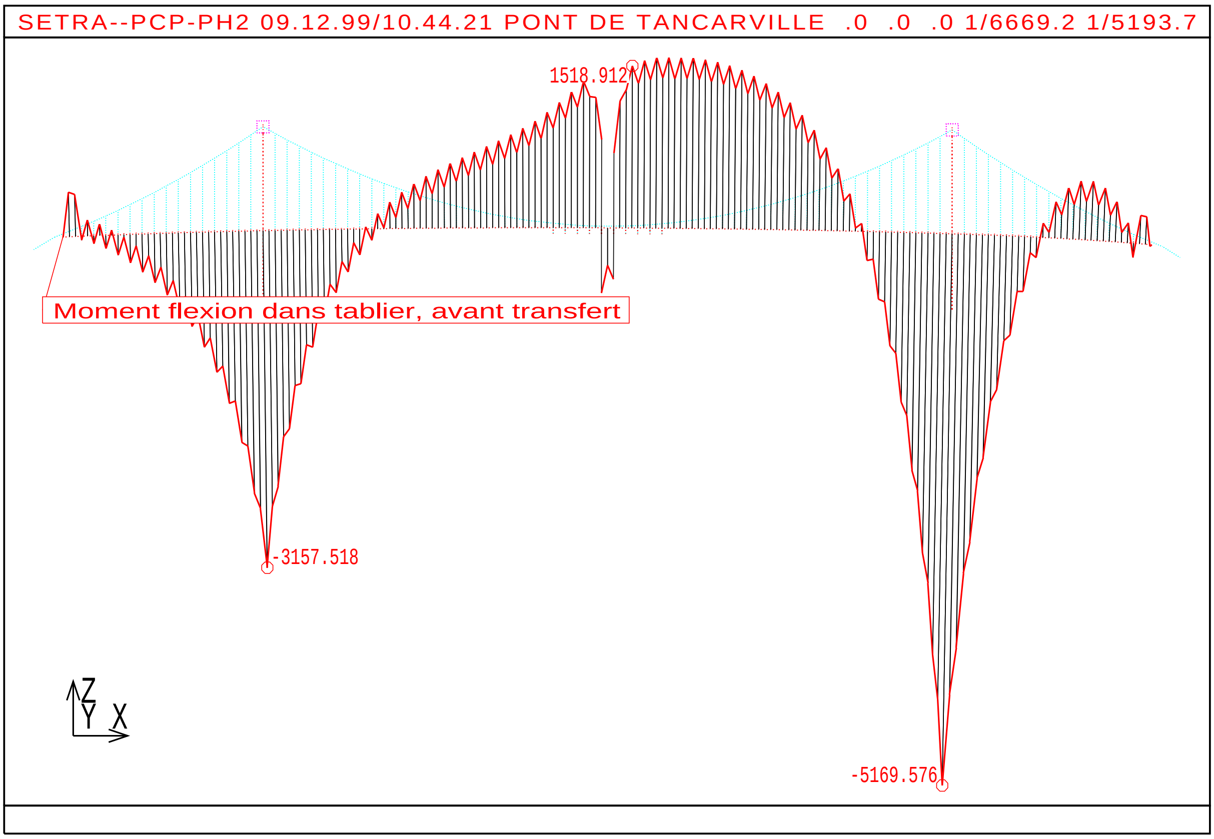


Figure 0.15 - General mechanical model, results on reducing active structure

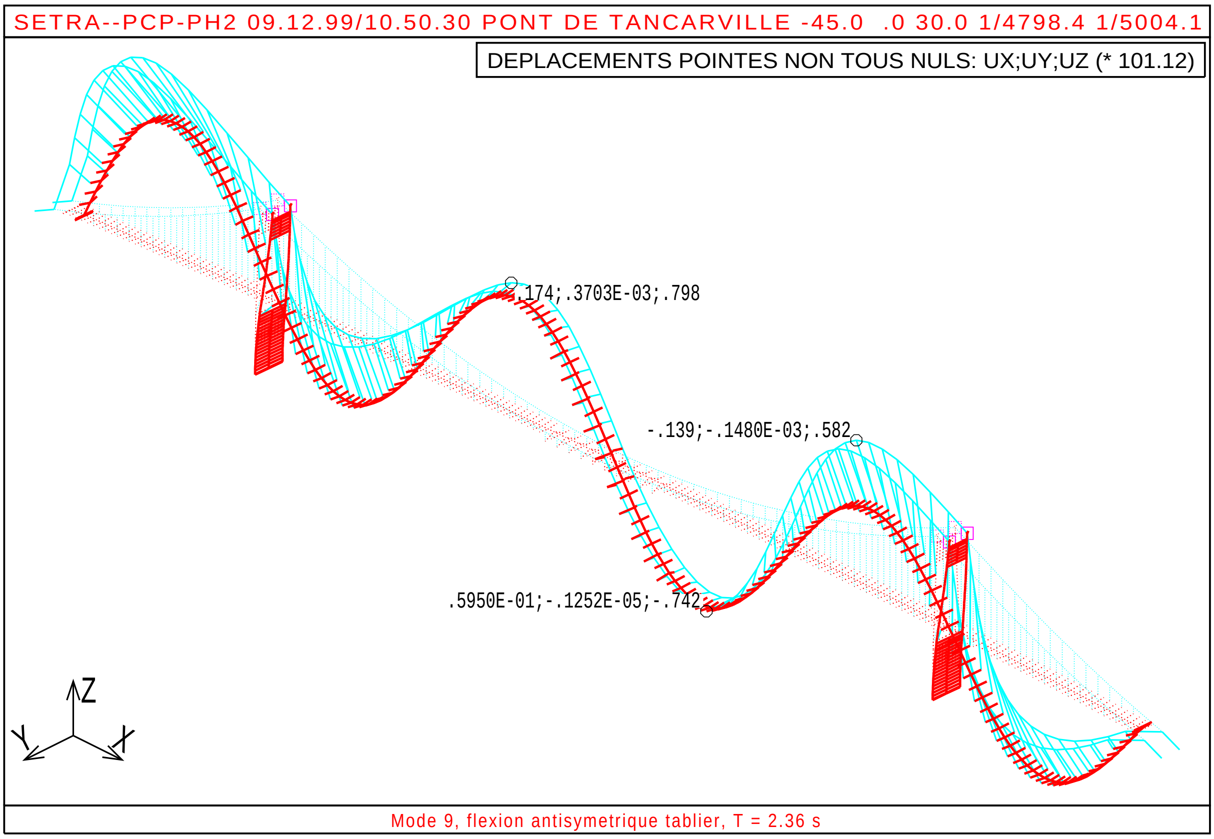


Figure 0.16 - General mechanical model, active part and vibration mode

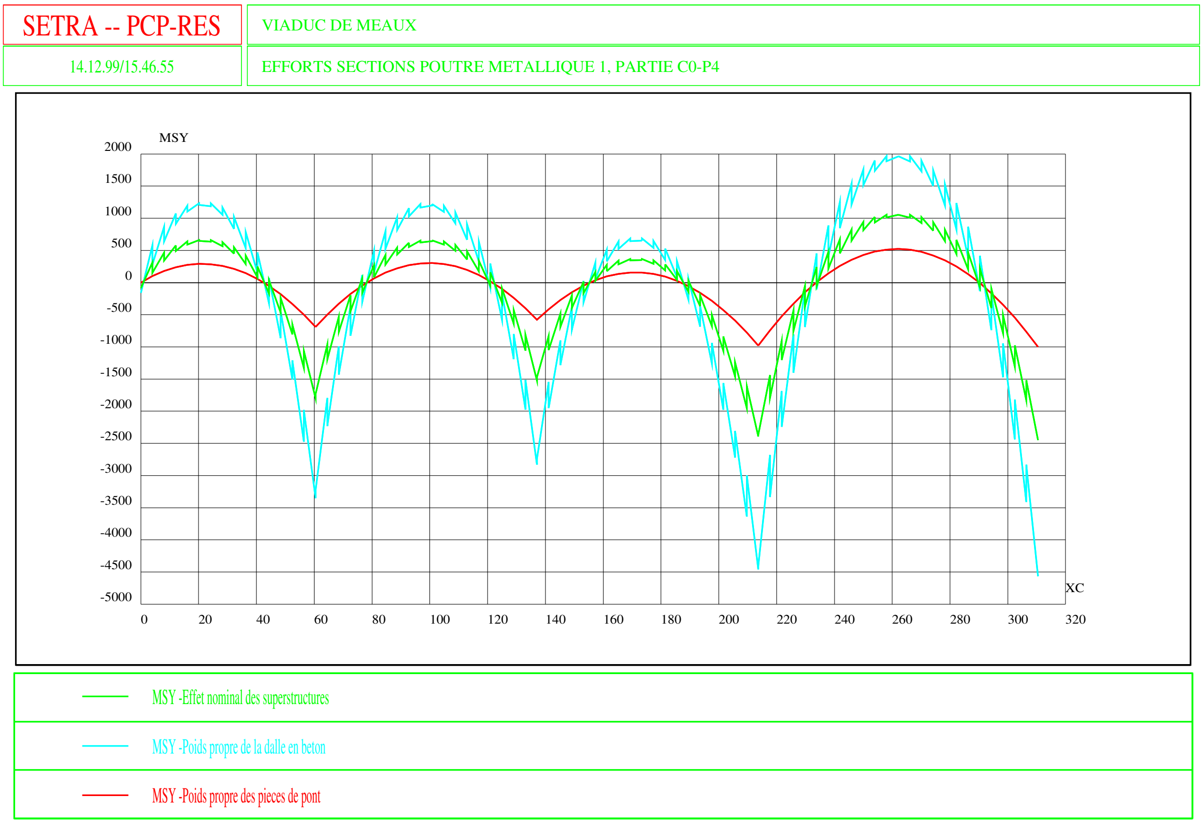


Figure 0.17 - Results shown separately, graph 1 (standard appearance)

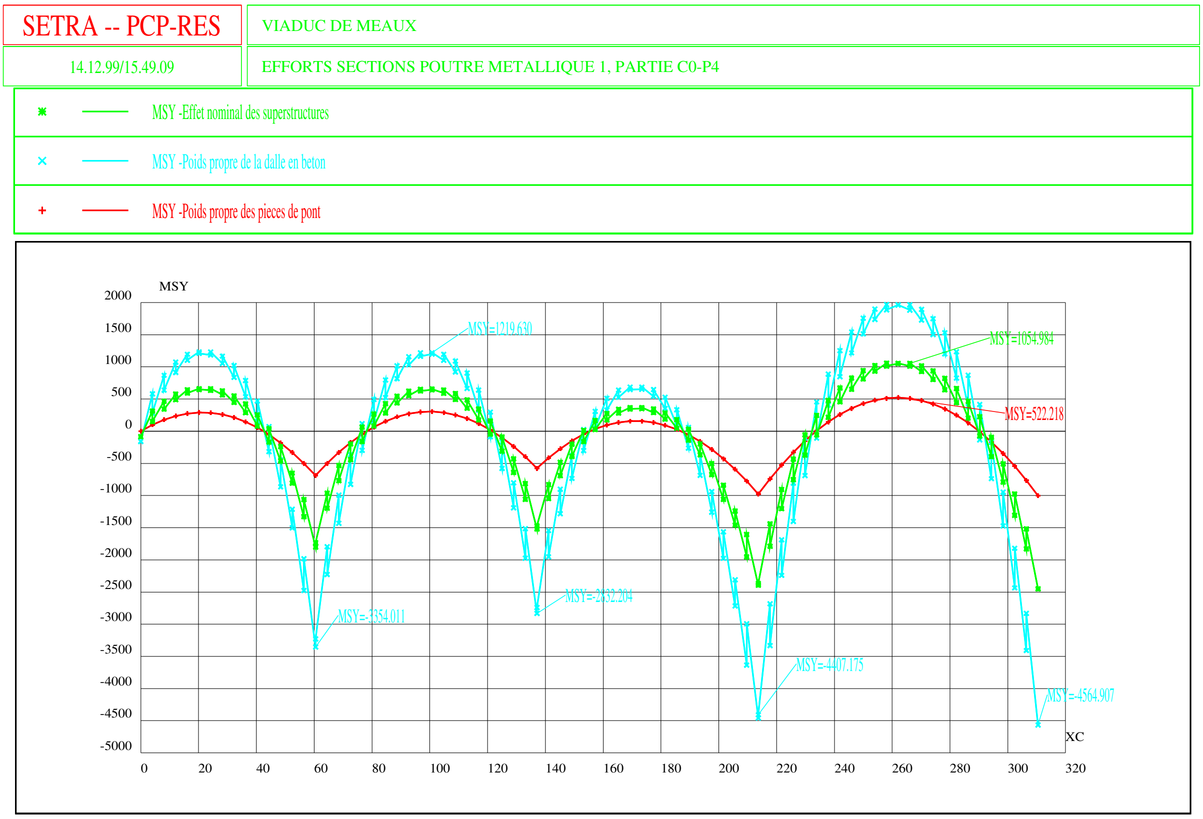


Figure 0.18 - Results shown separately, graph 2 (custom appearance)

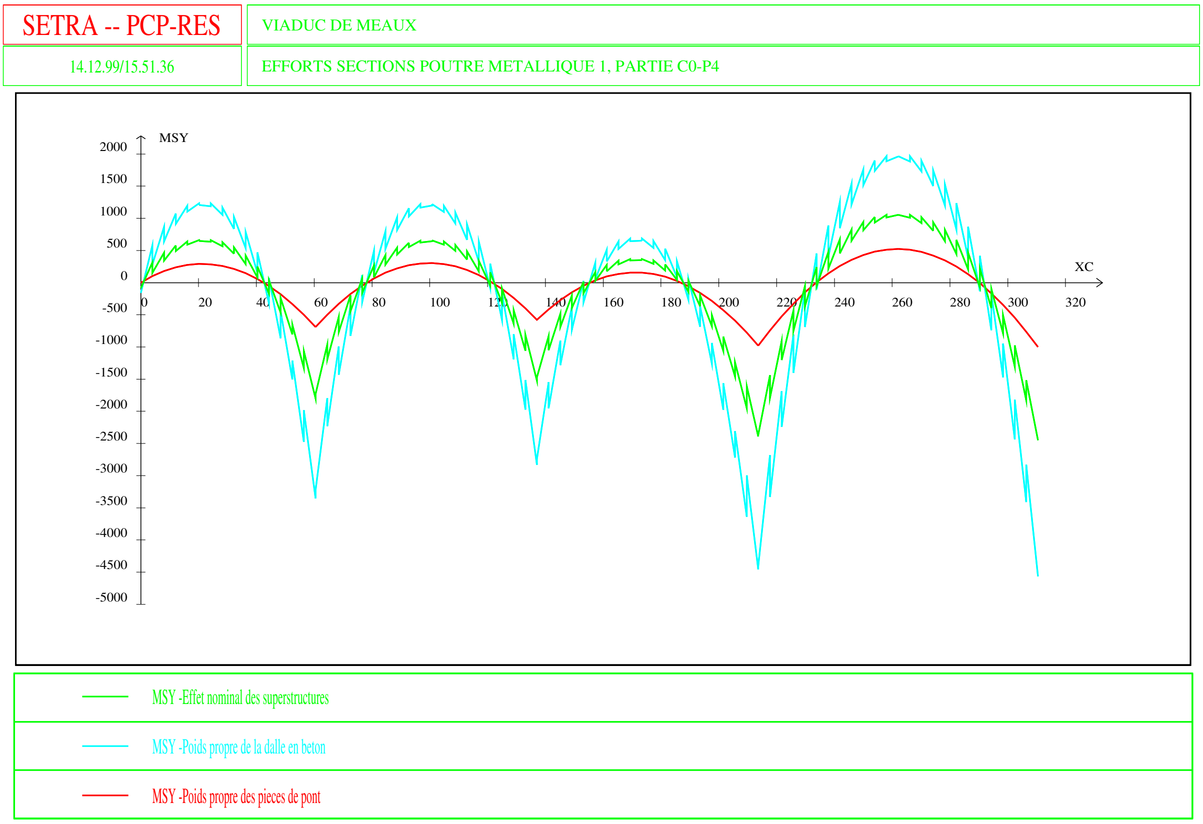
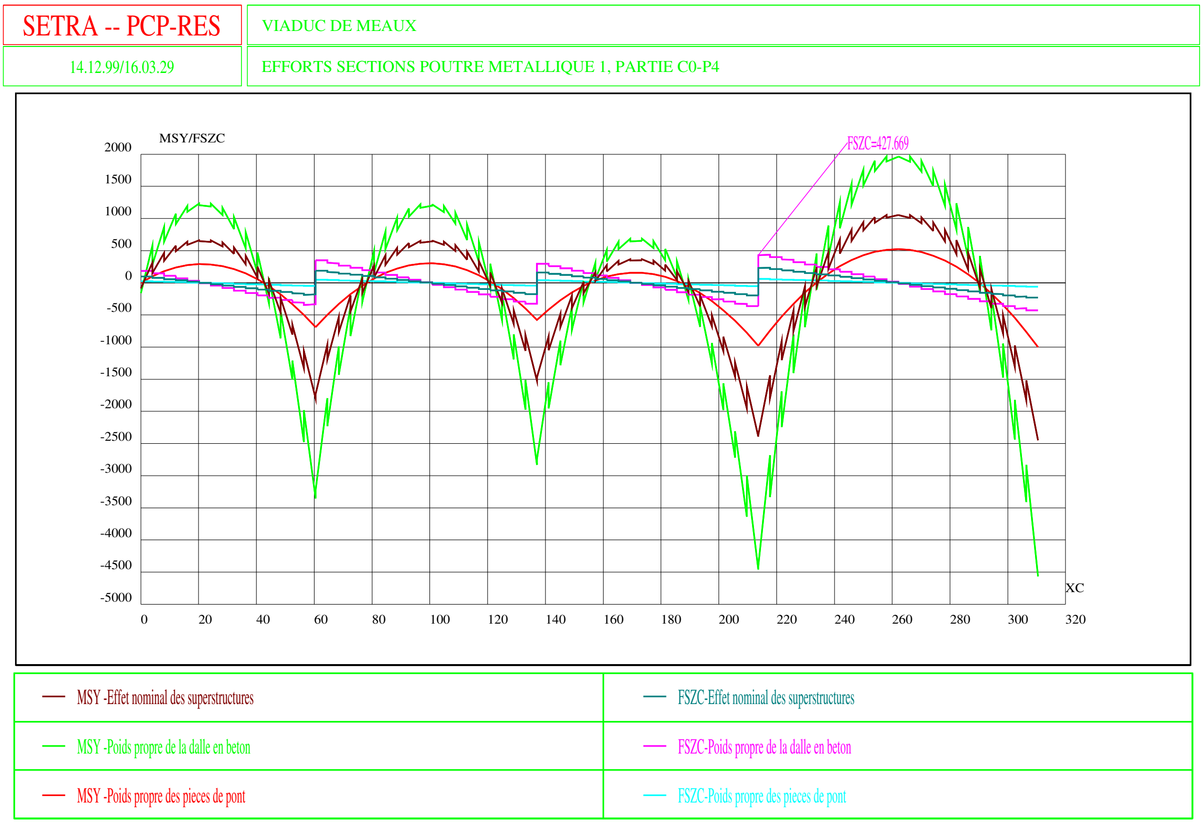


Figure 0.19 - Results shown separately, graph 3 (custom appearance)



SETRA -- PCP-RES

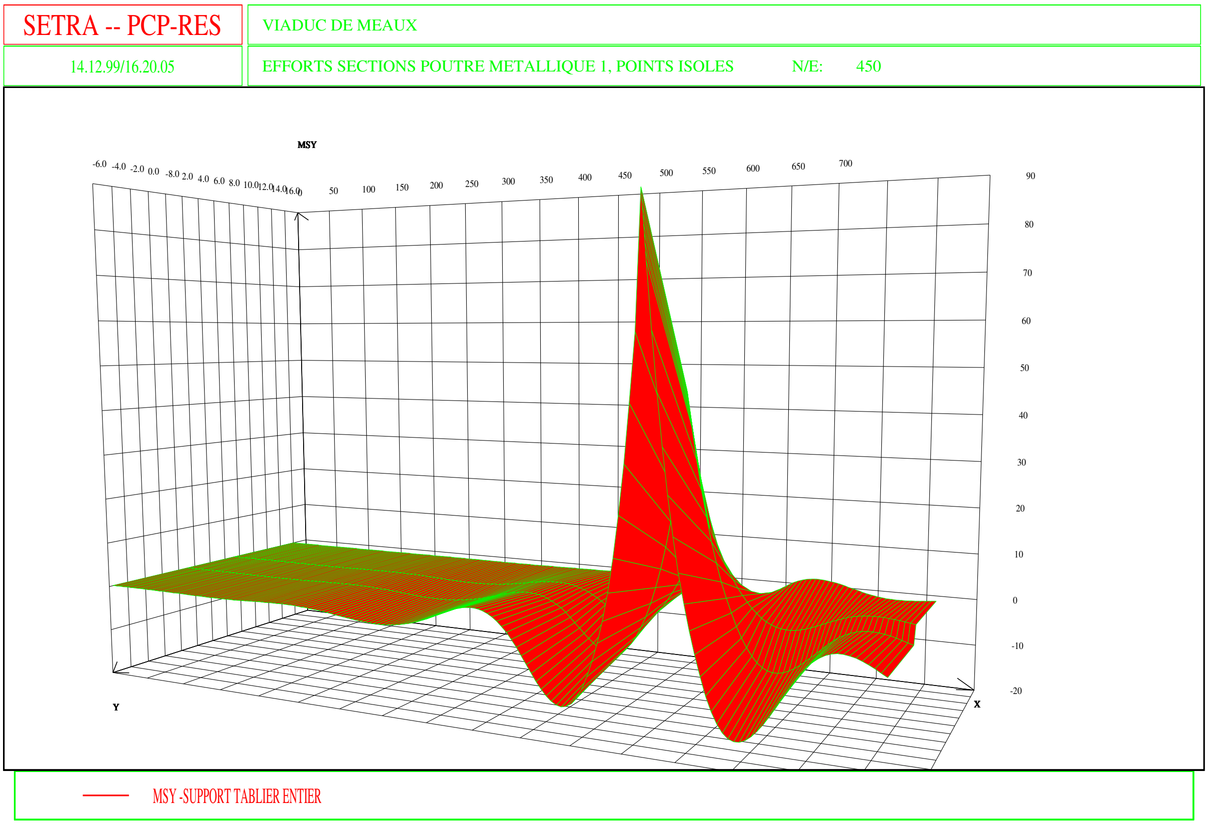
VIADUC DE MEAUX

14.12.99/16.03.29

EFFORTS SECTIONS POUTRE METALLIQUE 1, PARTIE C0-P4

MSY -Effet nominal des superstructures	FSZC-Effet nominal des superstructures
MSY -Poids propre de la dalle en beton	FSZC-Poids propre de la dalle en beton
MSY -Poids propre des pieces de pont	FSZC-Poids propre des pieces de pont

Figure 0.20 - Results shown separately, paragraph 4 (standard appearance)



SETRA -- PCP-RES

VIADUC DE MEAUX

14.12.99/16.20.05

EFFORTS SECTIONS POUTRE METALLIQUE 1, POINTS ISOLES N/E: 450

MSY -SUPPORT TABLIER ENTIER

Figure 0.21 - Results shown separately, area of influence, perspective

Chapter 1

General conventions

In this chapter, the term "beam" means a spatial beam.

CONTENTS

Contents

1.1 – COORDINATE REFERENCE FRAMES

All coordinate reference frames used by the PCP software are orthonormal and direct; the angles are counted positively in the trigonometric direction.

This section only discusses the reference frames used by several modules. Other reference frames, whose use is restricted to one module (ENV and DYN in particular), are described in the relevant chapters (9 and 11).

Global reference frame

This is the main reference frame, denoted "OXYZ", with respect to which the whole structure to be studied, and some loads to apply to it, are identified.

Its OZ axis should be vertical and ascending to allow the proper management of gravitational forces.

Beam reference frame

Denoted "$O_bX_bY_bZ_b$", it can be attached to one or more beams, to facilitate the definition.

It is mandatory to define a beam in a particular reference frame, whose O_bZ_b axis is not vertical, when the tangent to its mean fiber remains vertical or "temporarily passes" through this direction.

Generic reference frame

It is the reference frame denoted "$o_ix_iy_iz_i$", originating from the current point of rank i of the beam reference fiber, and in which is described one of its cross sections (generic section).

Section reference frame

The mean fiber of a beam is considered instead of the centers of gravity, G_i, from generic sections.

The axis G_ix_s of the reference frame of the cross section of rank i, of a beam, is tangent to the mean fiber and has the same direction of travel.

The central principal axes of the generic section (supposed close to those of the cross section), appropriately reoriented, complete the triad denoted "$G_ix_sy_sz_s$".

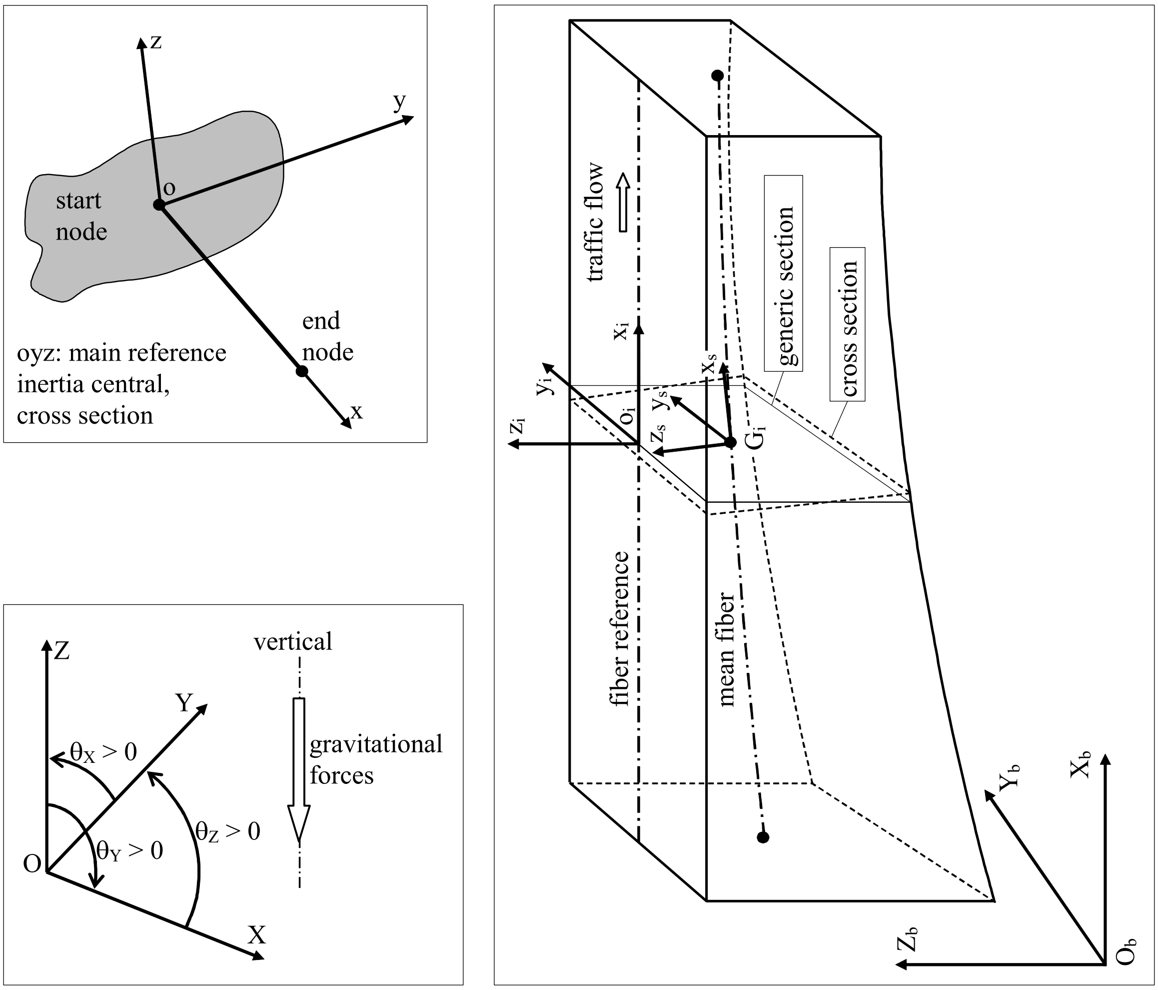
Forces calculated at the origins and ends of beam elements can be projected onto the reference frames of the corresponding cross sections and provide a basis for calculating the normal and tangential stresses in those sections.

Element reference frame

The mean line of beam elements is determined automatically; that of the non-beam elements is generally provided.

The ox axis of the local reference frame of a standardized prismatic element is carried by its mean line and its direction of travel is from the origin node to the end node.

The central principal axes of the cross section which is attached to it complete the triad denoted "oxyz" (for beam elements, a mean reference frame between the origin and end sections is used).



(sign conventions of rotations in GLOBAL reference apply to all types of references)

OXYZ: GLOBAL reference $O_pX_pY_pZ_p$: attached reference to one or several beams

$o_ix_iy_iz_i$: GENERIC reference i section, beam (i.e. G_i) $G_ix_sy_sz_s$: SECTION reference i section, beam

oxyz: ELEMENT reference, standardized prismatic element (from beam cut-up or off-beam)s

1Figure 1.1 - Coordinate reference frames

1.2 - CHANGING REFERENCE FRAMES

The spatial rotation that leads from a reference frame triad, denoted "PXYZ", to a "local" triad, denoted "Pxyz", consists of three elementary rotations, θ_1, θ_2 and θ_3, applied in this order and along the axes below.

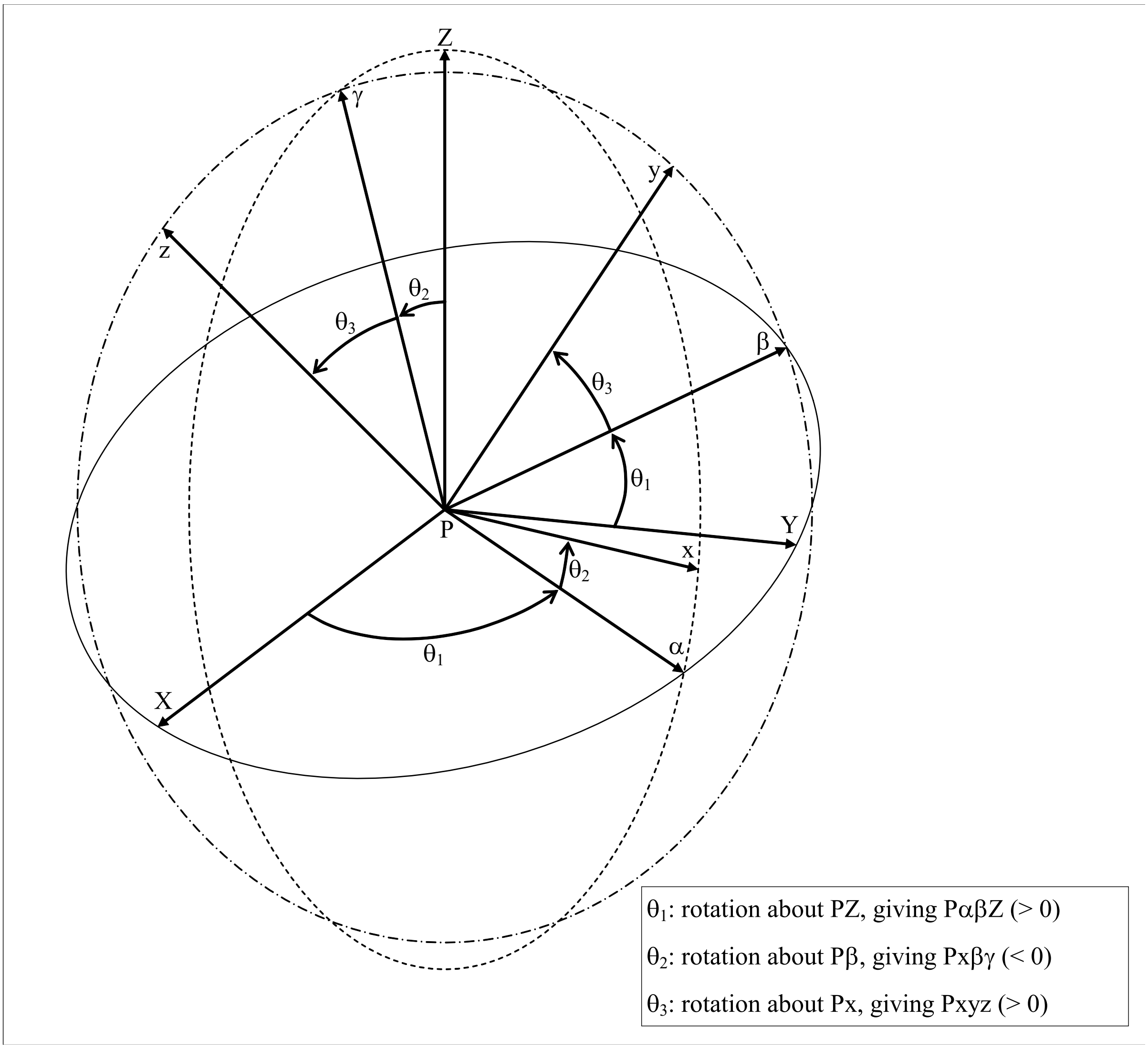


Figure 1.2 - Changing reference frames, triple normalized rotation

Remarks

These three rotations are performed in "reverse alphabetical" order:

- around PZ;
- Then, around the "new PY axis" (Pβ);
- finally, around the "new PX axis" (Px).

In the diagram above, θ_2 is chosen negative for better clarity.

1.3 - RESULTS, COMPONENTS

The symbols described below and the numbers of the corresponding components are used by some calculation modules and in some tables of results that they produce.

type of results	**C - 1**	**C - 2**	**C - 3**	**C - 4**	**C - 5**	**C - 6**
REACTIONS studied in the local reference frame of the supports (3 forces, 3 moments)	RFX	RFY	RFZ	RMX	RMY	RMZ
DISPLACEMENT of nodes studied in the global reference frame (3 displacements in the true sense, 3 rotations)	DEX	DEY	DEZ	ROX	ROY	ROZ
FORCES exercised by the nodes on the elements, studied in the local reference frames of beam or non-beam ELEMENTS, at their origins or ends (3 forces, 3 moments)	FEX	FEY	FEZ	MEX	MEY	MEZ
FORCES studied in the local reference frames of beam CROSS SECTIONS, at the origins or ends of their elements (3 forces, 3 moments, "left" torsors)	FSX	FSY	FSZ	MSX	MSY	MSZ
NORMAL STRESSES studied at the calculation points of cross sections of the beam elements, at the origins or ends of their elements	SIG					
TANGENTIAL and NORMAL STRESSES, as above	TAU	SIG				

Table 1.1 - Results, main components

Concomitant component marking symbols are obtained by adding the suffix 'C' to those of the main components (e.g.: TAUC and IACS for concomitant tangential and normal stresses).

1.4 - UNITS

The table below brings together the units used to write PCP software data and the exceptions and comments relating to them.

Type	Unit	Exceptions, comments
Length	meter (m)	Fits well with the orders of magnitude of the coordinates in structures of "conventional" dimensions, and certain geometric tolerances fixed in absolute mode
Mass	At the choice of the user: ton (t) or Mkg or otherwise.	The UNITES command in the DYN module (see Chapter 11) indicates the selected unit. Ton is used by default.
Force	At the choice of the user: kdaN, MN or otherwise.	The UNITES command in the PH3 module (see Chapter 8) indicates the selected unit. KdaN is used by default.
Time	day (d)	To process rheological phenomena
Angle	degree (°)	Please note, angular deflection coefficients of standard cables in the GE1 module (see Chapter 3) are expressed in degrees^{-1} The coefficients of stiffness or flexibility of the standard supports and standard articulations of the PH1 module (see Chapter 6) use radians (rd) The deformations of supports and articulations and distortions of elements required by the PH3 module (see Chapter 8) use radians (rd) The angles of orientation of the standard supports and standard articulations at the time of their placement by the PH3 module (see Chapter 8), however, are entered in degrees (°) The angular derivatives of the aerodynamic characteristics of the elements necessary for the DYN module (see Chapter 11) use radians (rd) Node rotations calculated and edited by the PH3 module are expressed in radians (rd)
Frequency	hertz (Hz)	
Time Period	second (s)	
Temperature	degrees Celsius (°C)	

Table 1.2 – Units

Please note that the CDS command of the ETU module (see Chapter 12) uses PCP software input units and CDS software output units (in the commands generated).

Chapter 2

Command language

CONTENTS

Contents

2.1 - AUTHORIZED CHARACTERS

The line length of a data file is limited to 80 characters.

The use of characters not listed in the table below (in particular: @, £, % and accented letters) can lead to decoding and transcription errors during certain file transfers.

Characters	Use
Letters a-z, except e and d Letters A-Z, except E and D	Names, keywords, character strings, comments Lowercase, which translates as uppercase upon acquisition, is indistinguishable from uppercase, outside strings
Letters e and E	Exponents of certain single-precision real constants, names, keywords, character strings, comments
Letters d and D	Exponents of double-precision real constants, names, keywords, character strings, comments
Numbers 0-9	Names, keywords, numerical constants, character strings, comments
Underscore _	Names, keywords, character strings, comments
Space	Lexical element separator Character strings, comments Blank lines are allowed in unlimited numbers
Comma ,	Expression separator in a list of expressions Character strings, comments
Dollar $ or Hash #	Marks the beginning of a comment
Period .	Separates the integer portion from the fractional part of a real constant Character strings, comments
Double asterisk **	Raise to power operator in expressions Character strings, comments
Asterisk * divided by /	Multiplication and division operators in expressions, repetition operator (asterisk) Character strings, comments
Plus + minus –	Unary operators for numeric constants, variables, functions or expressions Addition or concatenation operators, and subtraction in expressions Character strings, comments
Opening bracket (closing bracket)	Sub-expression, phrase and expression list delimiters Character strings, comments
Less than < greater than >	Assignment "box" and directed list delimiters Character strings, comments
Single quote ' double quote "	Character string delimiters Character strings, comments

Table 2.1 - Allowed characters and use

2.2 - BASIC LEXICAL ELEMENTS

No lexical element can be written on more than one data line.

Integer constant (ENTIER)

A non-empty list of numbers, preceded or not by a sign and including 11 characters maximum; it must belong to the interval: $[-2^{31}, 2^{31}-1]$.

Examples:

```
1      007     -10000     +500
```

Real single precision constant (REEL_SIMPLE)

Can be:

- a basic real constant;
- an integer constant followed by an exponent in single precision;
- or a basic real constant followed by an exponent in single precision.

The basic real constant comprises an optional sign, followed by an integer part (list of numbers), a decimal point, and a fractional part (list of numbers). The integer part may be omitted, or the fractional part, but not both; however, it is advisable not to use this possibility to improve the readability of data.

Its single precision exponent includes the letter e or E followed by an integer constant (signed or not).

Its total length must not exceed 14 characters and the exponent is limited to 25 in absolute value; its number of reliable digits is 6 (most common machine dependent values).

Examples:

```
+0125          .125         -12.50       +125E1       125E+001     -125e-01
+12.5E-01      -12.5E+1     -12.5e2      .125E0       -125.E3
```

Real double precision constant (REEL_DOUBLE)

Replace the letter e (or E) by d (or D) in the definition of the real single-precision constant.

It allows "precise" calculations to be carried out, however, the generated data will be acquired and stored in single precision, therefore sometimes truncated.

Its total length must not exceed 24 characters and the exponent is limited to 286 in absolute value; its number of reliable digits is 15 (most common machine dependent values).

Examples:

```
3.14159265D0           $ this value of PI keeps its precision
1D0    1.5D-2    .5d-6
```

Name (NOM)

A series of one to 8 characters (maximum length), composed of letters and/or numbers and/or underscores, beginning with a letter, and in which successive occurrences of the underscore, and a terminal underscore are not allowed.

Furthermore, reserved words: **LIS**, **DIM, POUR**, **BRISER**, **PASSER**, **SI**, **SINON**, **LIRE**, **RETOURNER**, **E**, **D**, **A**, **INC**, **DIV** and **REP**, are not used as names.

Examples:

```
$ correct names
BETON_T    BETON_L    C12_13S
STRUSERV   Fleau_2    Sect_105   CAB100
C_102g     equ_1_2    Appb       ARTI_Bl
Sargin     BPELELU    KARMAN     KAIMAL

$ incorrect names
STRUCSERV  $ too long
1D10G      $ does not start with a letter
D__10G     $ contains two consecutive underlined blank spaces
D.10G      $ contains an unauthorized full stop
D10G_      $ ends with an underlined blank space
```

Keyword

It is constructed, decoded and checked as a name, but its length is limited only by that of the data line.

When it exceeds four characters (in the wording of a command descriptor), only its first four characters are required to be coded for it to be recognized as a predefined command language keyword. Keywords of length less than or equal to four characters must be entirely coded (see also section 2.11).

Examples:

```
SECTION    Sect   CARActeristiques    CARA        CARACT
Tendre     TEND   ACTIVER             ACTI
Z_MESURE   Z_ME   LOG                 W           AXE
```

Character string (STRING)

A series of characters enclosed in single or double quotes, in which the included single or double quotes are allowed if they are distinct from the beginning or the end.

Its length is only limited by that of the data line.

Examples:

```
   $ there is no apostrophe included
   '*** ETAT DE L OUVRAGE A SA MISE EN SERVICE ***'
   $ the included apostrophe is authorized because contained within speech
marks
   "*** ETAT DE L'OUVRAGE A SA MISE EN SERVICE ***"
```

Comment

All characters following the first dollar ($) or the first hash (#) found on a line (included), up to the end of line, are ignored when decoding and can be used to insert explanatory comments (see also section 2.10).

Examples:

```
$ start of construction
$ ------------------------
DATE   0   $ indicates the switch to
           $ CALCUL RHEOLOGIQUE FIN
PLACER  APPUIS  2  $ piles P2 and P3
```

End of command delimiter.

This is the semicolon character (;) (see also section 2.10).

Examples:

```
SURCHARGES  TITRE  'TEST'  VERIFICATION;
```

Lexical elements separator

Besides the comma, operators and delimiters of expressions, lists of expressions or directed lists (**, *, /, +, −, (,), < and >), the following characters are used as separators of lexical items:

- spaces in any number, not included in a string or comment;
- command end delimiter not included in a string or comment;
- newline.

It is mandatory to separate a name (or keyword) from another name (or keyword) or a numerical constant, and a numerical constant from another numerical constant or name (or keyword).

2.3 - DATA FILE

A data file consists exclusively of any comments and commands.

Comments

A comment is a basic lexical item ignored by the application. It can be introduced at the start of the line, or within the line, if it respects the integrity of its "usefulness".

Its characters are not analyzed as a series of lexical elements, which in particular allows inclusion of apostrophes, unpaired quotes or brackets, or lexically nonconforming "names".

Examples:

```
TENDRE  CABLES  2 $ tensioning of the temporary cables
CAB_1  CAB_2

$ status of the construction when put into service
ETAT  100

$ installation of the piles 1A and 1B
PLACER  APPUIS  2
APPU_1  105  6*0.0   APPU_2  205  6*0.0

$ the insertion of the dollar too early introduces an error
<VA = 30.0$> initialisation of the variable VA
```

Commands

Commands, consisting of other types of basic lexical elements (names, keywords, numeric constants, character strings), are generally used to define an entity (broadly defined) or action to be taken.

They always start with one or more keyword identifiers, followed by other possible lexical items of various types, according to their imposed syntax.

Optional end delimiter commands

They are used by the GE1, GE2, GE4, MC1, PH1 and PH3 modules, and include:

- a header that holds the first keywords, followed by zero or more other lexical elements, all introduced on the same line;
- a possible list of associated lexical elements, coded on the following lines, free of format constraints;
- an optional end delimiter (;).

Examples:

```
$ commands without list
TITRE 'ETUDE DU POUSSAGE'
TENSIONS  GROUPE  2

$ commands with list
ACTIVER  ELEMENTS  10
105 110 115 120 125 130 135 140 145 150
```

The end of command delimiter can separate one command from another, without passing to the next line, but the line break between command header and the list (if it exists) is still required.

Examples:

```
$ this example is not ideally legible
CALCULER  CONTRAINTES ; CALCULER  EXTREMAS ; DATE  0
SUSPENDRE  EXTREMAS ; TENDRE  CABLES  2
CAB_1  CAB_2 ; ACTIVER  ELEMENTS  3
105 107 109 ; EDITER
```

Mandatory end delimiter commands

They are employed by the ENV, DYN and ETU modules and the TRAFFIC sub-module and include:

- the first keywords, followed by zero or more lexical elements;
- an end delimiter (;).

No line break is imposed within this type of command; it may be written on several lines and a line may contain several.

Examples:

```
$ the semicolons are appended to the commands as a reminder
$ that they are part of them. They also act as separators
PCPETU; RAPPELER  TERMINAL; FIN;

$ command (of the ENV module) on several lines
$ in order to separate the logical blocks
ETUDE  11
'TABLIER NORD, REACTIONS D''APPUIS'
REACTIONS
NOEUDS  1  36  86  121;

$ command (of the ENV module) on a single line
$ (the length of the character chain permits this)
ACTION  100 'A(l)'  SUPPORT  103  TRAFIC  AL;

$ another command (of the DYN module) on a single line
$ (still quite legible)
ADMITTANCE  MODALE  'ADMITTANCE MOYENNE'  MOYENNE  0.75;
```

Numerical constants

When an ENTIER parameter is called in the label of a command, only an ENTIER constant, variable, or expression can be introduced.

A real parameter can be introduced as an ENTIER, REEL_SIMPLE or REEL_DOUBLE value (constant, variable or expression).

Examples:

```
    $ certain real parameters are replaced by integers
    MATERIAU  BETON_1  3
    3  14  2
    3.6E6  0.20  2.50
    1.0E-5 3600000  1  0.95

    $ error. The number of elements to be activated must always be an
integer
    ACTIVER  ELEMENTS  POIDS  2.0
    103 104
```

2.4 - COMMAND DESCRIPTIONS

The command descriptions meet certain presentation rules detailed below.

Character attributes

Keywords are shown in UPPERCASE characters (not bold and not accented), in their complete form.

Although they are recognized by their prefix of four characters maximum, it is advisable to label them entirely, to enhance readability.

In the corresponding comments, the word "option" is used instead of "keyword".

Some UNDERLINED keywords are the default options; for example, for this part of the label:

$$\textbf{MASSES} \quad \textbf{titre\_masses} \quad \left\{ \begin{array}{ll} \underline{\text{ABSOLUES}} & \dots \\ \text{RELATIVES} & \dots \end{array} \right\}$$

these three entries are equivalent:

```
MASSES  'DEVIATEURS'  ...
MASSES  'DEVIATEURS'  ABSOLUES  ...
MASS  'DEVIATEURS'  ABSO  ...
```

The names that must necessarily be coded in full are presented in **BOLD UPPERCASE**; for example:

$$\textbf{TAULIMITES} \quad \dots \quad \left\{ \begin{array}{l} \textbf{CHALOS} \\ \textbf{BPELELS} \\ \textbf{BPELELU} \end{array} \right\} \quad \dots$$

The parameters of all types are presented in unaccented non-italic lowercase Latin characters, and/or non-italic lowercase Greek characters; examples:

nom_arti **no_element** θ_1 $d\theta_z$

Italic lowercase characters representing parts of command descriptions are to be substituted due to their excessive clutter.

Special characters

When a parameter is in brackets, or when multiple parameters are in brackets and separated by commas, these are predefined variable names, to be labeled as such, with brackets (and commas); examples:

(type_voies_0, degres_0) **(bande_n_char)**

The square brackets include optional entities (keywords, parameters, or more elaborate entities); examples:

POUSSER [AUTOMATIQUEMENT] STRUCTURES …

TENDRE CABLES [NONINJECTES] [nb_cables]

$$\mathbf{[}\left\{\begin{matrix}\langle \text{no\_noeud}\rangle_{\text{nb\_noeuds}} \\ \langle \text{no\_element}\rangle_{\text{nb\_elements}}\end{matrix}\right\}\mathbf{]}$$

[POUTRES nb_poutres

$$\langle \text{no\_poutre}\rangle_{\text{nb\_poutres}}\ \mathbf{]}$$

The "boxes" include lists of entities to be repeated a number of times indicated by a parameter name, attached as a subscript; example:

$$\left\langle \text{x\_point}\quad \text{y\_point}\quad \text{z\_point}\quad \theta_x\quad \theta_y\quad \theta_z \right\rangle_{\text{nb\_points}}$$

The nesting of several boxes is possible, as well as the insertion of insignificant line breaks to save space; examples:

$$\left\langle \langle \text{z\_point}\rangle_{\text{nb\_files}} \right\rangle_{\text{nb\_points}}$$

$$\begin{array}{l}\left\langle \text{ETUDE}\quad \text{no\_domaine} \right. \\ \\ \text{[} \\ \left\langle \text{COMPOSANTE}\quad \text{no\_cpp}\quad \text{[CONCOMITANTES}\quad \langle \text{no\_cpc}\rangle_{\text{nb\_cpc}}\ \text{]} \right\rangle_{\text{nb\_cpp}}\ \text{]} \left.\right\rangle_{\text{nb\_domaines}}\end{array}$$

Single braces express an exclusive choice, between each of the entities they cover; for example, this part of the wording:

$$\left\{\begin{array}{l}\text{REACTIONS} \\ \\ \text{EFFORTS}\quad \left\{\begin{matrix}\text{ELEMENTS} \\ \text{SECTIONS}\end{matrix}\right\} \\ \text{CONTRAINTES}\quad \left\{\begin{matrix}\text{NORMALES} \\ \text{TANGENTES}\end{matrix}\right\}\end{array}\right\}$$

can result in the following entries (exhaustive list):

```
      REACTIONS
   or EFFORTS  ELEMENTS
   or EFFORTS  SECTIONS
```

```
    or  CONTRAINTES  NORMALES
    or  CONTRAINTES  TANGENTES
```

When a single asterisk precedes the braces, any combination of entities encompassed is allowed, with a maximum of one instance of each entity; their order of introduction is indifferent; for example, this part of the wording:

$$*\left\{\begin{matrix}\text{DEPLACEMENTS} \\ \text{REACTIONS} \\ \text{EFFORTS}\end{matrix}\right\}$$

can result in the following entries (non-exhaustive list):

```
        DEPLACEMENTS
    or  EFFORTS
    or  DEPLACEMENTS  REACTIONS
    or  EFFORTS  DEPLACEMENTS
    or  REACTIONS  EFFORTS  DEPLACEMENTS
```

When a double asterisk precedes the braces, any combination of entities encompassed is allowed, with no limitation on the number of their occurrences, the order of their introduction is also indifferent; for example, this part of the wording:

$$**\left\{\begin{matrix}\text{SECTION} & \text{nom\_section} \\ \text{POUTRES} & \langle\text{no\_poutre}\rangle_{\text{nb\_poutres}}\end{matrix}\right\}$$

can result in the following entries (non-exhaustive list):

```
        SECTION  S1  SECTION  S2
    or  POUTRES  101 102
    or  SECTION  S1  POUTRES  101  102  SECTION  S2
```

When a plus sign precedes the braces, one copy of each of the non-optional entities encompassed, and a maximum of one copy of each of the optional entities encompassed must be introduced, in a non-imposed global order; for example, this part of the wording (amended):

$$+\left\{\begin{matrix}\text{MASSE\_VOLUMIQUE\_AIR} \quad \rho \\ \text{Z\_SOL} \quad \text{zs} \\ [\text{RUGOSITE\_SITE} \quad z_0]\end{matrix}\right\}$$

can result in the following entries (exhaustive list):

```
        MASSE_VOLUMIQUE_AIR  0.00122  Z_SOL  10.0
    or  Z_SOL  10.0  MASSE_VOLUMIQUE_AIR  0.00122
    or  MASSE_VOLUMIQUE_AIR  0.00122  Z_SOL  10.0  RUGOSITE  SITE  5.0
    or  MASSE_VOLUMIQUE_AIR  0.00122  RUGOSITE  SITE  5.0  Z_SOL  10.0
    or  Z_SOL  10.0  MASSE_VOLUMIQUE_AIR  0.00122  RUGOSITE  SITE  5.0
    or  Z_SOL  10.0  RUGOSITE  SITE  5.0  MASSE_VOLUMIQUE_AIR  0.00122
    or  RUGOSITE  SITE  5.0  MASSE_VOLUMIQUE_AIR  0.00122  Z_SOL  10.0
    or  RUGOSITE  SITE  5.0  Z_SOL  10.0  MASSE_VOLUMIQUE_AIR  0.00122
```

Where an entity of a group in braces undergoes a shift to the right, it should be viewed as a continuation of the previous one (left-aligned), and not as a separate entity.

It would be the same if a special character ([, ⟨) could only find its "symmetrical" entity on the line following it; for example, these two entries are equivalent:

$$\left\{\begin{array}{l}\text{DIVISION\_CHARGE\_MAX}\quad\text{nb\_fois}\\ \text{MODE}\quad\text{no\_mode}\\ \quad\text{[STRUCTURE}\quad\text{nom\_structure]}\quad\text{FACTEUR}\quad\text{facteur}\end{array}\right\}$$

$$\left\{\begin{array}{l}\text{DIVISION\_CHARGE\_MAX}\quad\text{nb\_fois}\\ \text{MODE}\quad\text{no\_mode}\quad\text{[STRUCTURE}\quad\text{nom\_structure]}\quad\text{FACTEUR}\quad\text{facteur}\end{array}\right\}$$

Chapter 2 (continued)

Pseudo-Programming

CONTENTS

The pseudo-programming language proposed here replaces the language proposed in earlier versions of PCP (before 6.10) with which it is fully compatible but to which it adds a number of features.

The proposed pseudo-language is very close to that of ST1. It allows nested loops, conditional breaks, file reads. It specifies the lists and arrays. The last paragraph of this chapter gives the main differences between the PCP and ST1 languages.

Contents

2.5 – VARIABLES

Presentation

A variable is a name associated with a number or a string.

Syntax

variable_name = value

< variable_name = value >

with:

- value: constant, parenthesized expression, array element, or string.

Specifications

- A variable name must be different from a list or array name.
- A variable name must not be a reserved word of the language **LIS**, **DIM,** **POUR**, **BRISER**, **PASSER**, **SI**, **SINON**, **LIRE**, **RETOURNER**, **E**, **D**, **A**, **INC**, **DIV** and **REP**.
- A variable name cannot be later used to refer to a list or an array.
- A variable is created by an assignment command.
- It may be set equal to an expression.
- It can receive several values successively.
- It is referenced by its name in parentheses or without.
- It can be referenced in expressions.
- Any referenced variable must have been defined.
- Any single reference to a variable returns the value of the variable.

Examples

```
<PI = 3.14159>              $ REEL_SIMPLE type variable
<X = 100>                   $ ENTIER type variable
<PI_D = 3.14159265D0>       $ REEL_DOUBLE type variable
<X = (4/2)>                 $ variable X is redefined
(X)            ==> 2
(X**X)         ==> 4
<Y = 3>
<Z = X*Y>      ==> Error the expression is not placed between brackets
<Z = (X*Y)>
(Z)            ==> 6
<A = 125.50>                $ Frequent error. The name A
                            $ is forbidden as a variable name
PI = 3.14159                $ REEL SIMPLE type variable
PI_D = 3.14159265D0         $ REEL_DOUBLE type variable
X = (4/2)                   $ variable X is redefined
X              ==> 2
(X**X)         ==> 4
Z = (X*Y)
Z =  X*Y       ==> Error the expression is not placed between brackets
Z              ==> 6
CH = "chaine quelconque" $ Chain
CH             ==> "chaine quelconque"
(CH)           ==> chaine quelconque
```

Some PCP commands need a title as a parameter as a string. This title can be entered directly or thanks to a variable including a string :

```
name ='bridge support subsidence'
TITRE name
```

2.6 – ARRAYS

Presentation

An array is a sequence of digital values referenceable by an index.

Syntax

DIM Tu(n) [= v_1,v_2,v_3,v_4,,v_i,,,v_n]

DIM Tu(n) [= v]

DIM Tb(nl,nc) [= v_1,v_2,v_3,v_4,,v_i,,,v_{nl*nc}]

DIM Tb(nl,nc) [=v]

with:

- Tu: name of the one-dimensional array.
- Tb: name of the two-dimensional array.
- n: number of values in the one-dimensional array.
- nl: number of values in the two-dimensional array.
- nc: number of columns in the one-dimensional array.
- v_i: constant numerical value, expression, list name, array, variable or incrementor, all separated by commas and, possibly, spaces.
- v: constant numerical value, expression or variable assigned to all the array elements. By default, the assigned value is zero.

Specifications

- An array name must be different from a variable or array name.
- An array name must not be a reserved word of the language **LIS**, **DIM, POUR**, **BRISER**, **PASSER**, **SI**, **SINON**, **LIRE**, **RETOURNER**, **E**, **D**, **A**, **INC**, **DIV** and **REP**.
- An array name cannot be later used to refer to a variable or a list.
- An array is created by the DIM instruction. The array name and the opening parenthesis must be contiguous.
- An array can receive values when created by the DIM instruction. If only one value is assigned, it is assigned to all the array elements. If multiple values are set, all the values in the array must be filled.
- An array can generally receive values after its creation: all the values in the array must then filled.
- An array can be assigned to another array: the two arrays must have the same dimensions.
- For a two-dimensional array, the input order of the values is as follows: all the values of the column 1, followed by the values of the column 2, and so on.

- Each array cell can receive a value using the index in parentheses: T(i)=v or T(i,j)=v. The array name and the opening bracket must be contiguous.
- It is referenced by its name followed by the index in parentheses: T(i) or T(i,j) with no spaces between the name and the opening parenthesis.
- Any referenced array element must have received a value.
- Array dimensions can be constants, variables or entire expressions.
- Array indices can be constants, variables or entire expressions.
- Any reference to an array as operand without indices concerns the entire array: matrix or vector calculation.
- Any single reference to an array without indices returns the contents of the array ordered column by column, i.e. by successively adding 1 to the cardinality of the column index and by varying each value of the column index by adding 1 to the cardinality of the row index.

Examples

```
DIM t(2,3)= 1
t                  ==> 1 1 1 1 1 1
DIM t(2,3)= 1 a 6
t                  ==> 1 2 3 4 5 6
t(2,1)             ==> 2
t(2,2)=5
t                  ==> 1 2 3 5 5 6
t = 7 a 12
t                  ==> 7 8 9 10 11 12
t = 7 a 13         ==> Error too many assigned values
t = 1              ==> Error not enough assigned values

dim ta(2,2)=1
dim tb(2,2)=2
dim tc(2,2)=3
ta=tb
ta                 ==> 2 2 2 2
ta=(tc)
ta                 ==> 3 3 3 3
# One-dimensional table
k=0
n=10
DIM t(n)
POUR  i = 1 a n << k=(k+i) t(i)=k >>

# These expressions are equivalent
ACTIVER ELEMENTS n
POUR  i = 1 a n << t(i) >>

ACTIVER ELEMENTS n
T

ACTIVER ELEMENTS n
(t)



# Two-dimensional table
n=3
m = 2
DIM t(n,m)
POUR  i = 1 a n
<<
   POUR j = 1 a m << t(i,j)= (i+j) >>
>>
```

```
# These expressions are equivalent
t          => 2 3 4 3 4 5

POUR  j = 1 a m
<<
   POUR i = 1 a n << t(i,j) >>
>>
           => 2 3 4 3 4 5

# Example of dimensions with the same values as previously
DIM t(n,m)= 2,3,4,3,4,5
```

2.7 – SCALAR FUNCTIONS

Presentation

The array below describes the available features, usable individually or in various expressions but always in parentheses. Their arguments can be constants, variables, elements of arrays or expressions. The **CPTR** argument must be CHAINE type. For a function to be recognized, the opening parenthesis must be contiguous with the name of the function.

Function	Description	Type of result
MAX(x,y)	Maximum value of x and y	Type of x and y
MIN(x,y)	Minimum value of x and y	Type of x and y
SIN(x)	Sine of x; x in radians	REEL, according to the type of x
COS(x)	Cosine of x; x in radians	REEL, according to the type of x
TAN(x)	Tangent of x; x in radians, of different from ±π/2 to π close	REEL, according to the type of x
ASIN(x)	Arc sine of x; $-1.0 \leq x \leq 1.0$; $-\pi/2 \leq$ **ASIN**(x) $\leq \pi/2$	REEL, according to the type of x
ACOS(x)	Arc cosine of x; $-1.0 \leq x \leq 1.0$; $0.0 \leq$ **ACOS**(x) $\leq \pi$	REEL, according to the type of x
ATAN(x)	Arc tangent of x; $-\pi/2 <$ **ATAN**(x) $< \pi/2$	REEL, according to the type of x
LOG(x)	Natural logarithm of x; $x> 0.0$	REEL, according to the type of x
SINH(x)	Hyperbolic sine of x; x in radians	REEL, according to the type of x
COSH(x)	Hyperbolic cosine of x; x in radians	REEL, according to the type of x
TANH(x)	Hyperbolic tangent of x; x in radians, differing by ±π/2 to within π	REEL, according to the type of x
EXP(x)	Exponential of x	REEL, according to the type of x
ABS(x)	Absolute value of x	Type of x
INT(x)	Integer part of x	ENTIER
INT(x)	Integer part of x	ENTIER
ARR(x)	The nearest integer value	ENTIER
REEL(x)	Conversion of x, to a real single precision value	REEL_SIMPLE
DBLE(x)	Conversion of x, to a real double precision value	REEL_DOUBLE
CPTR('x')	Counter representing the current number of definitions of the variable x, in a directed list	ENTIER
XCLOT(a,r,s) **YCLOT**(a,r,s)	Abscissa and ordinate of a point on a clothoid of parameter a, whose radius carried by the y-axis is r, and the curvilinear abscissa from the coordinate origin is s	Type of greater precision parameter

Table 2.2 - Functions

Examples

```
<PI = 3.14159>
(TAN(PI/4.0))               ==> 0.9999987
(ACOS(COS(0.5D0))           ==> 0.5D0
(ABS(-1))                   ==> 1
(REEL(1))                   ==> 1.0
(ENT(2.5D0))                ==> 2
(XCLOT(1.0, 10.0, 0.5))     ==> 0.4982317
```

2.8 – VECTOR OR MATRIX FUNCTIONS

Presentation

The array below describes the available features, usable individually or in various expressions but always in parentheses. Their arguments can be arrays or array expressions. One-dimensional arrays are called **Vectors** and two-dimensional arrays **Matrices**.

Function	Description	Type of result
NEU(x)	Euclidean norm of the vector x or matrix x	Real
NIN(x)	Infinite norm of the vector x or matrix x	Real
DET(x)	Determinant of the square matrix x	Real
RNG(x)	Rank of the square matrix x	Integer
INV(x)	Inverse of the square matrix x	Matrix
TRA(x)	Transpose of the square matrix x	Array
DIA(x)	Diagonal of the square matrix x	Vector
SOL(a,b)	Solution x of ax = b, with a(n,n) square matrix and b(n,m)	Matrix x(n,m)
PVE(x,y)	Cross product x.y of two vectors with three components	Vector
PSC(x,y)	Scalar product x.y of two vectors of the same dimensions	Real

Table 2.3 - Functions

Examples

```
dim t (3,5)= 1 a 15
Neu(t) 35.2136337233

Nin(t) 15

dim t(3,3)= 1 a 9
Dia(t) 1 5 9
Tra(t) 1 4 7 2 5 8 3 6 9

Dim ta (3)=1,0,0
Dim tb (3)=0,1,0
Pve(ta,tb) 0,0,1
Psc(ta,tb) 0

dim tc(3,3)= -3, -1, 1, 5, 2, -1, 6, 2, -1
Det(tc)1
rng(tc)3
Inv(tc) 0 -1 1 1 3 -2 2 3.33066907388d-016 1

dim td(4,4) = 2,-4,4,0,1,-2,1,-3,0,3,-2,-12,4,-7,8,-1
dim te(4,2) = 2,-9,2,2,2,-9,2,2
sol(td,te) 3 4 -1 -2 3 4 -1 -2
```

2.9 – SCALAR EXPRESSIONS

Formation rules

An expression is constructed from operators and operands, and placed in parentheses.

Symbol	Syntax; effects
+	+ operand; no effect on numeric operand
–	– operand; sign change of a digital operand
+	operand + operand; addition of two numeric operands
+ or &	operand + operand or operand & operand; concatenation of two strings of characters, or a character string with an integer, to form a name
–	operand – operand; subtraction of the second operand from the first (numeric)
*	operand * operand; multiplication of two numeric operands
/	operand / operand; division of the first operand by the second (numeric)
**	operand ** operand; raising of the first operand to the power of the second (numeric)

Table 2.4 - Operators

An operand can be a primary, factor, term, or expression, as in the array below.

Operand	Definition
expression	expression + term or expression – term or + term or – term
term	term * factor or term / factor or factor
factor	primary ** primary or primary
primary	constant or variable_name or function_name (expression [, expression, expression, …]) or (expression)
constant	numeric constant type ENTIER, REEL_SIMPLE or REEL_DOUBLE or character string

Table 2.5 - Expression formation rules

This structure is extracted from the FORTRAN V writing rules (which defines the factor as a possible combination of a primary and a factor, which is the only difference).

The separators of lexical items (especially spaces and ends of lines) are allowed in any numbers in the expressions and ignored; giving in particular the possibility of ventilating an expression and/or writing it over multiple lines.

Examples

```
$ simple expressions based on constants only
(2 + 3)       ==> 5                   (2 - 3)          ==> -1
(2 * 3)       ==> 6                   (((2 ** 3)))     ==> 8
(2 / 3)       ==> 0                   (’SECT’ + ’AR’)  ==> SECTAR
(’SECT’ + 6)  ==> SECT6               (2 + 7)          ==> 9
(’SECT’ & 6)  ==> SECT6

$ simple expressions based on constants and variables
(XA = 6>   <XB = 7>   <XC = 30>   <XD = 3.0>
(XA+XB)    ==> 13
(XC*XA)    ==> 180
(XD**2.0)  ==> 9.00000E0

$ simple expressions based on constants, variables and functions
<XA = 2.0D0>   <XB = 4.0D0>
<PI = 3.14159265D0>
(2.0D0*SIN(PI/6.0D0))    ==> 1.00000D0
(3.0*ABS(XA-XB))         ==> 6.00000D0
(LOG(XA) + LOG(XB**2))   ==> 3.46574D0

$ "precedence" errors
(4**+2)   (5*+5)   (6/-3)
```

Evaluation rules

The various components of an expression are evaluated by traversing the cells of Array 2.5 from the bottom up, i.e., in order of priority:

- the primaries, the most nested sub-expressions, functions and variables;
- factors;
- terms.

In other words, the order of processing is:

- parentheses, functions and variables;
- operators ** (of the highest priority);
- operators * and / (of the same level of intermediate priority);
- operators + and – (of the same level of lowest priority).

And operators with the same priority level are evaluated from left to right.

Examples:

```
(2 * 2 + 2)                     ==> 6
(2 * 3 / 6 + 5 * 3)             ==> 16
(2 ** 3 + 6)                    ==> 14
(2 * (3 + 6) - 7 * (4 / 3))     ==> 11
(7 * 4 / 3)                     ==> 9
```

Type conversion rules

An operation between two operands of the same type (numeric or CHAINE) gives a result of the type of these operands.

The concatenation of an ENTIER type operand with a CHAINE type operand, gives a CHAINE type result.

The results of evaluations of CHAINE type expressions are converted into lexical elements of NOM type and controlled as such.

Examples:

```
(2 * 3.0)              ==> 6.00000E0
(2 + 3.0D0)            ==> 5.00000D0
(2.0 * 3.0D0)          ==> 6.00000D0
(2 + 3.0 * 4.0D0)      ==> 14.0000D0
('CABL' + 101)         ==> CABL101  $ correct name
('CAB' + 'FL' + 'D01') ==> CABFLD01 $ correct name
```

Some more elaborate expressions:

```
<X = 3.0>
<Y = ((2*X*X+3*X+4)/(3*X-5))>
(Y)                                         ==> 7.750000E0
<ALFAD = 12.5)
<PI = 3.14159>
<ALFAR = (ALFAD*PI/180.0)>
<V = (3*(COS(ALFAR)**2-2*SIN(ALFAR)))>
(V)                                         ==> 1.560826E0
<KSI = 0.5D0>
<W = (LOG((1-KSI**2)/(2.5285*KSI-1.0))**0.5)>
(EXP(W))                                    ==> 2.7769704796232D0
```

Some PCP commands need a title as a parameter as a string. This title can be entered directly or thanks to a variable including a string :

```
name =('bridge support subsidence' + 1)
TITRE name

I = 1
name =('bridge support subsidence' + I)
TITRE name
```

2.10 – VECTOR OR MATRIX EXPRESSIONS

Formation rules

An expression is constructed from operators and operands, and placed in parentheses. The operands are vectors or matrices unless otherwise stated.

Symbol	Syntax; effects
+	+ operand; no effect on numeric operand
–	– operand; change of sign of each term of the operand
+	operand + operand; addition of two numeric operands
+	operand + scalar; addition of a scalar to every term of the operand
–	operand – operand; subtraction of the second operand from the first
–	operand – scalar; subtraction of a scalar from every term of the operand
∗	operand ∗ operand; multiplication of two operands a(n,p) =b(n,m)*b(m,p)
∗	vector ∗ vector; scalar product between two vectors of the same dimension
∗	operand ∗ scalar; multiplication by the scalar of the first operand
/	operand / operand; solves ax=b, i.e. x =a/b with a(n,n), b(n,m) and x(n,m)
/	operand / scalar; division of each term of the first operand by the scalar

Table 2.6 - Vector or matrix operators

The precedence rules are the same as for the scalar expressions.

Examples

```
Dim ta(2,2)=1
Dim tb(2,2)=2

(ta + tb)          ==> 3  3  3  3
(ta - tb)          ==> -1  -1  -1  -1
(ta * tb)          ==> 4  4  4  4
(tb + 2)           ==> 1  1  1  1
(tb - 2)           ==> 0  0  0  0
(tb * 2)           ==> 4  4  4  4

dim td(4,4) = 2,-4,4,0,1,-2,1,-3,0,3,-2,-12,4,-7,8,-1
dim te(4,2) = 2,-9,2,2,2,-9,2,2

(te/td)            ==> 3  4  -1  -2  3  4  -1  -2

dim tr(4,2)
tr = (te/td)
tr                 ==> 3  4  -1  -2  3  4  -1  -2
```

2.11 - EXPRESSION LISTS

Presentation

A series of expressions or terms separated by commas, placed in parentheses.

Syntax

(expression [,expression, expression, …])

Specifications

- Each term can be a numeric expression, array, an array element, a list name, a variable or a string.
- Double quotes and single quotes delimiting character strings are deleted in the string produced.

Examples

```
<X = 2.0>     <Y = 3.0>
(3*X*X+4.0*X+5.0, SIN(0.5*X),
4.0*LOG(Y+3.0*X+2.0)            ==> 25.0 0.841471 9.59158
(2.0*X+1.5763, 3.0*Y**X)        ==> 5.57630 27.0
("Chaine1",5,"Chaine2")         ==> Chaine1 5 Chaine2
("Chaine1 5 Chaine2")           ==> Chaine1 5 Chaine2
```

2.12 – EXPRESSION NAME LISTS

Presentation

A list name is a name associated with a sequence of numeric values or strings.

Syntax

list_name = $v_1,v_2,v_3,v_4,,v_i,,,v_n$

< list_name = $v_1,v_2,v_3,v_4,,v_i,,,v_n$ >

list_name = ($v_1,v_2,v_3,v_4,,v_i,,,v_n$)

< list_name = ($v_1,v_2,v_3,v_4,,v_i,,,v_n$) >

list_name = empty

LIS list_name

where:

- v_i: constant numerical value, expression, string, list name, array, array element, variable or incrementor all separated by commas.
- empty: empty list.

Specifications

- A list name must be different from a keyword of the language, variable or array.
- A list name cannot be later used to refer to a variable or an array.
- All values are separated by the character "," and possibly spaces. They can be enclosed in parentheses.
- Double quotes and single quotes delimiting strings are removed if the list is parenthesized totally or in part.
- A list is coded over several lines by placing a "," character as the last character of intermediate lines.
- A list is created by an assignment command. It can successively receive several series of values, the new replacing the old. An empty list can be assigned to it.
- It can reference other lists and reference itself to be completed.
- It is referenced by name.
- Any isolated reference returns the contents of the list.
- Any referenced list must have been defined possibly empty.

Examples

```
# Example of overload
Ls1=1,2,5 a 8
<Ls1=1 a 10>

# Example of concatenation
Ls2=(1 a 3,x1,4,I)
pour j=1 a 5 << ls2=ls2,j >>

# list on several lines
ll=1 a 3,
5 a 10,
20 a 30
ACTIVER ELEMENTS n
Ll

# Bracketed dequoted list
<Ls3 = ("cable","materiau",10,"element")>
Ls3                 => cable materiau 10 element

#Non-bracketed list: non-dequoted.
<Ls4 = "cable","materiau",10,"element"> Ls4
                    => "cable" "materiau" 10 "element"

# Empty list
Lis Ls5

# Empty list
Ls5 = vide

# Completed empty list
Ls5 = Ls5, 1 a 10
```

2.13 - REPEATERS

Presentation

It allows repetition of a series of data.

Syntax

n * x

where:

- n: ENTIER element indicating the number of times the element x is to be repeated, positive;
- x: numeric element (ENTIER, REEL_SIMPLE or REEL_DOUBLE).

Specifications

- n and x can be constants or expressions.
- No separators (of lexical items) are allowed in a repeater, apart from spaces; which means writing over multiple lines is forbidden.

Examples

```
4*3            ==> 3 3 3 3
5 * 20         ==> 20 20 20 20 20
<A = 4>
<B = 7.0>
(A)*(B+5.0)    ==> 12.0 12.0 12.0 12.0
```

2.14 - INCREMENTORS

Presentation

This is to generate a sequence of numeric elements (ENTIER, REEL_SIMPLE or REEL_DOUBLE) by successive additions of a unit increment, provided or calculated.

Syntax

x **A** y [from x, to y (included), with a unit increment] {1}

x **A** y **INC** z [from x, to y (included), with an increment z] {2}

x **A** y **DIV** n [from x, to y (included), with n intervals] {3}

x **REP** n **INC** z [from x, repeat n times: add z increment] {4}

where:

- x, y: different elements of the same numeric type (ENTIER, REEL_SIMPLE or REEL_DOUBLE);
- z: non-zero value of the increment, which should be the same type as x (and y if appropriate); its sign must be that of y–x; in the form {1}, the absolute value of z is taken as 1, 1.0E0 or 1.0D0 depending on the type of x and y;
- n: integer number of equal intervals to generate, between the terminals x and y for form {3}; number of increments z to add successively to x for form {4}; must be, in both cases, greater than or equal to 1.

A, INC, DIV and **REP** are reserved words of the command language that cannot be used as variable names nor as names for the application.

x, y, z and n may be constants or expressions.

No separators (of lexical items) are allowed in an incrementor, apart from spaces; which means writing over multiple lines is forbidden.

Generation rules

The list of elements generated by the form "x **A** y" is as follows:

$$x \quad x+1 \quad \cdots \quad x+i \quad \cdots \quad y \qquad si:\ y>x \tag{2.1}$$

$$x \quad x-1 \quad \cdots \quad x-i \quad \cdots \quad y \qquad si:\ y<x \tag{2.2}$$

And if *ade* is the penultimate element generated:

$$|y-1| \le |ade| < |y-1.0\mathrm{E}-6| \tag{2.3}$$

Examples:

```
1 A 10            ==> 1 2 3 4 5 6 7 8 9 10
(2*5) A (2+3)     ==> 10 9 8 7 6 5
1 A -1            ==> 1 0 -1
6.0 A 8.5         ==> 6.0 7.0 8.0 8.5
5.0 A 8.000001    ==> 5.00000  6.00000  7.00000  8.000001
5.0 A 8.00001     ==> 5.00000  6.00000  7.00000  8.00000  8.00001
1 A 1             ==> error, equal bounds
1 A 10.0          ==> error, non-homogeneous types
```

The list of elements generated by the form "x **A** y **INC** z" is as follows:

$$x \quad x+z \quad \cdots \quad x+i\cdot z \quad \cdots \quad y \tag{2.4}$$

And if *ade* is the penultimate element generated:

$$|y-z| \le |ade| < \left| y - \frac{z}{1.0\mathrm{E}6} \right| \tag{2.5}$$

Examples:

```
<SECTD = 1>   <SECTF = 5>
(SECTD) A (SECTF) INC 2        ==> 1 3 5
1 A 6 INC 2                    ==> 1 3 5 6
1.0 A 60.0 INC 10.0            ==> 1.0 11.0 21.0 31.0 41.0 51.0 60.0
80 A 60 INC -6                 ==> 80 74 68 62 60
15.0 A 5.0 INC -2.0            ==> 15.0 13.0 11.0 9.0 7.0 5.0
1 A 5 INC 2.0                  ==> error, non-homogeneous types
5.0 A 5.0                      ==> error, equal bounds
1 A 6 INC 0                    ==> error, nul increment
```

The list of elements generated by the form "x **A** y **DIV** n" is as follows:

$$x \quad x+\frac{y-x}{n} \quad \cdots \quad x+i\cdot\frac{y-x}{n} \quad \cdots \quad y \tag{2.6}$$

When x and y are of type ENTIER, it is the whole part of $\frac{y-x}{n}$ which is taken as increment.

Examples:

```
1 A 2 DIV 1             ==> 1 2
0 A (10+9*10) DIV 2     ==> 0 50 100
1.0 A 2.0 DIV 4         ==> 1.00 1.25 1.50 1.75 2.00
10 A 0 DIV 10           ==> 10 9 8 7 6 5 4 3 2 1 0
1 A 20 DIV 5            ==> 1 4 7 10 13 16 19 20
1 A 2.0 DIV 1           ==> error, non-homogeneous types
0 A 100 DIV 2.0         ==> error, the number of intervals is a real
                        number
1 A 2 DIV 2             ==> error, calculated increment is a real
                        number aequal to zero
```

The list of elements generated by the form "x **REP** n **INC** z" is as follows:

$$x \quad x+z \quad \ldots \quad x+i\cdot z \quad \ldots \quad x+n\cdot z \tag{2.7}$$

Examples:

```
<CINQE = 5>
1 REP (CINQE) INC 2     ==> 1 3 5 7 9 11
1.0 REP 3 INC 1.2       ==> 1 2.2 3.4 4.6
1.0 REP 5.0 INC 2.0     ==> error, number of increments is a real
                        number
1.0 REP 3 INC 0.0       ==> error, nul increment
1.0 REP 3 INC 2         ==> error, non-homogeneous types
```

2.15 - DIRECTED LIST

Presentation

This is a list of entities enclosed in a "box" (< >) and repeated as many times as a variable receives values assigned in a bounding box (simple writing form of nesting level 1, {1}).

In a more elaborate form, a directed list can be contained in another directed list, by nesting, repeatedly (higher level nested writing form,{2}).

Syntax

liste_dirigee <==> <<liste_entites> nom_variable = liste_entites_numeriques> {1}

liste_dirigee <==> <liste_dirigee nom_variable = liste_entites_numeriques> {2}

where:

- liste_entites: list of constants, variables, repeaters, incrementors, separators and/or lists of expressions, defining arbitrary data types (ENTIER, REEL_SIMPLE, REEL_DOUBLE or CHAINE);
- nom_variable: name of an assigned variable;
- liste_entites_numeriques: list of constants, variables, repeaters, incrementors, separators and/or lists of expressions, of the same numeric type (ENTIER, REEL_SIMPLE or REEL_DOUBLE).

The maximum number of nested levels is configurable.

All variables assigned to a given nesting level can be invoked in entities in a lower level, therefore more internal.

Evaluation and generation rules

For a directed list of the form {2}, the variable receives the successive values of the list of numeric entities, liste_entites_numeriques, going from left to right, and the nested list, liste_dirigee, is translated for each of these values.

When the directed list yet to be evaluated is of the form {1}, the list of entities, liste_entites, is translated for each successive value taken by the variable.

This sequence of successive evaluations of the list of entities builds the data list defined by the nesting sequence of directed lists.

Thus the recurrent order of evaluation is:

- the least nested directed list;
- the corresponding list of numeric entities, going from left to right;
- the directed list of the next lower level, or the list of entities, going from left to right.

The separators of lexical items or expressions (line ends, commas or other) in the lists of numeric entities are accepted and considered as non-significant.

Those included in the lists of entities are accepted and considered as significant, and therefore transmitted to the application.

This is the case in particular for line ends, which, when inserted judiciously, can produce lists of values with a certain structure, which are therefore more easily verifiable.

Examples:

```
    <<4.0> X = 1 A 5>    ==> 4.0 4.0 4.0 4.0 4.0

    <<1 A 5> X = 1 A 2>  ==> 1 2 3 4 5 1 2 3 4 5

    <<  $ The carriage return inserted here is integrated in the data
generated
    1 A 5> X = 1 A 3>        ==> 1 2 3 4 5
                                 1 2 3 4 5
                                 1 2 3 4 5

    $ when X is not between brackets, it is considered as a name
    <<X 0 0> X = 1 A 5>      ==> X 0 0 X 0 0 X 0 0 X 0 0 X 0 0

    $ X between brackets is translated into its numerical equivalent
    <<(X) 0 0> X = 1 A 5>    ==> 1 0 0 2 0 0 3 0 0 4 0 0 5 0 0

    <<
    (COS(S/100.0), SIN(S/100.0), 0.05*S)>
    S = 0.0 A 100.0 DIV 5>          ==>   1.000     0.000     0.000
                                          0.9800666 0.1986693 1.000
                                          0.9210610 0.3894183 2.000
                                          0.8253356 0.5646425 3.000
                                          0.6967067 0.7173561 4.000
                                          0.5403023 0.8414710 5.000

    <T=3>
    $ 3*(T) is a repeater, whereas (3*T) is an expression
    <<(4*X*X+5*X+6) 0 0> X = 0 (1+1)
    (T, T+2) 3*(T) (3*T) A (5*T)>   ==> 6   0 0 32  0 0 57  0 0 131 0 0
                                        57  0 0 57  0 0 57  0 0 375 0 0
                                        456 0 0 545 0 0 642 0 0 747 0 0
                                        860 0 0,981 0 0

    $ Straight marker cable, 200 m long, with a point every 2 m
    $ using a DIV 100 instead of INC 2.0 avoids the division
    $ of the sections, to within one tolerance, at the abscissa 200.0
    FIBRE  REPERE  101  3
    <<
    (X) 5*0.0> X = 0.0 A 200.0 DIV 100>   ==> FIBRE REPERE 101 3
                                              0.00   0.0 0.0 0.0 0.0 0.0
                                              2.00   0.0 0.0 0.0 0.0 0.0
                                              ..........................
                                              200.00 0.0 0.0 0.0 0.0 0.0

    $ Marker fiber with 9 points, regularly spaced, on a circle with a
    $ radius of 100 m meters. Constant cant angle of 2 degrees
    $ The variable TR (angle in radians), a function of T (angle in
    $ is known at the innermost level of the nesting
    <PI = 3.141593>    <R = 100.0>    <XC = 86.6025>    <YC = -60.0>
    FIBRE REPERE 9 3
    <<<
    (XC+R*COS(TR), YC+R*SIN(TR), 0.0, T-90.0, 0.0, 2.0)> TR = (T*PI/180.0)>
    T = 150.0 A 30.0 DIV 8>  ==> FIBRE REPERE 9 3
                                  -0.53406E-04 -10.00000 0.0  60.000 0.0 2.0
                                  15.89181      10.71066 0.0  45.000 0.0 2.0
                                  36.60249      26.60254 0.0  30.000 0.0 2.0
                                  60.72058      36.59258 0.0  15.000 0.0 2.0
                                  86.60249      40.000   0.0   0.000 0.0 2.0
                                 112.4844       36.59259 0.0 -15.000 0.0 2.0
                                 136.6025       26.60255 0.0 -30.000 0.0 2.0
                                 157.3132       10.71069 0.0 -45.000 0.0 2.0
                                 173.2050      -10.000   0.0 -60.000 0.0 2.0
```

```
    $ series of six solid rectangular sections measuring 2 m in width
    $ whose height varies parabolically between 3 m and 2 m
    <<<
    SECTION  MASSIVE  (’SECT’ + CPTR(’X’)) 0
    4
    $ exterior contour and calculation points of the normal stresses
    -1.0 0.0 -1.0 (-H) 1.0 (-H) 1.0 0.0
    -1.0 0.0 -1.0 (-H) 1.0 (-H) 1.0 0.0
    0.0 0.0 5*1.0 3*0.0 0.0> $ fictive data
    H = (X*X/225.0+2.0)>
    X = -15.0 A 0.0 DIV 5>
           ==>    SECTION MASSIVE SECT1 0
                  4
                  -1.000 0.0 -1.000 -3.000 1.0 -3.000 1.0 0.0
                  -1.000 0.0 -1.000 -3.000 1.0 -3.000 1.0 0.0
                  0.0 0.0 1.0 1.0 1.0 1.0 1.0 0.0 0.0 0.0 0.0
                  SECTION MASSIVE SECT2 0
                  4
                  -1.000 0.0 -1.000 -2.64000 1.0 -2.64000 1.0 0.0
                  -1.000 0.0 -1.000 -2.64000 1.0 -2.64000 1.0 0.0
                  0.0 0.0 1.0 1.0 1.0 1.0 1.0 0.0 0.0 0.0 0.0
                  SECTION MASSIVE SECT3 0
                  4
                  -1.000 0.0 -1.000 -2.36000 1.0 -2.36000 1.0 0.0
                  -1.000 0.0 -1.000 -2.36000 1.0 -2.36000 1.0 0.0
                  0.0 0.0 1.0 1.0 1.0 1.0 1.0 0.0 0.0 0.0 0.0
                  SECTION MASSIVE SECT4 0
                  4
                  -1.000 0.0 -1.000 -2.16000 1.0 -2.16000 1.0 0.0
                  -1.000 0.0 -1.000 -2.16000 1.0 -2.16000 1.0 0.0
                  0.0 0.0 1.0 1.0 1.0 1.0 1.0 0.0 0.0 0.0 0.0
                  SECTION MASSIVE SECT5 0
                  4
                  -1.000 0.0 -1.000 -2.04000 1.0 -2.04000 1.0 0.0
                  -1.000 0.0 -1.000 -2.04000 1.0 -2.04000 1.0 0.0
                  0.0 0.0 1.0 1.0 1.0 1.0 1.0 0.0 0.0 0.0 0.0
                  SECTION MASSIVE SECT6 0
                  4
                  -1.000 0.0 -1.000 -2.000 1.0 -2.000 1.0 0.0
                  -1.000 0.0 -1.000 -2.000 1.0 -2.000 1.0 0.0
                  0.0 0.0 1.0 1.0 1.0 1.0 1.0 0.0 0.0 0.0 0.0

    $ list of incidences
    <<(I, I+1) > I = 1 A 10> ==> 1 2 2 3 3 4 4 5 5 6 6 7 7 8 8 9 9 10

    $ list of cable names
    <<
    (’CAB’ + IC + ’D’) > IC = 10 A 15>    ==> CAB10D
                                              CAB11D
                                              CAB12D
                                              CAB13D
                                              CAB14D
                                              CAB15D

    $ controlled list with three levels of nesting. The calculation
    $ is developed as three nested loops. The loop relative to z is the
outermost
    $ the output is reformatted for greater legibility
    $ this statement normally produces three values per line
    <<<<
        (x,y,z)> x = 1 A 2>
                  y = 1 A 3>
                    z = 1 A 4>
    ==> 1 1 1                                    $ z = 1, y = 1, x = 1
        2 1 1                                    $ z = 1, y = 1, x = 2
        1 2 1  2 2 1                             $ z = 1, y = 2, x = 1, 2
        1 3 1  2 3 1                             $ z = 1, y = 3, x = 1, 2
        1 1 2  2 1 2  1 2 2  2 2 2  1 3 2  2 3 2 $ z = 2, same
        1 1 3  2 1 3  1 2 3  2 2 3  1 3 3  2 3 3 $ z = 3, same
        1 1 4  2 1 4  1 2 4  2 2 4  1 3 4  2 3 4 $ z = 4, same
```

2.16 - CONCATENATOR

Presentation

This is to concatenate names and integer values outside expressions.

Syntax

name & name/v

Where:

- name: name type lexical term.
- v: integer type numeric value.

Specifications

- The numeric values v can be constants or expressions.
- The left hand side cannot be numeric.
- The "&" instructions can be combined.
- The difference between a concatenation in an expression and this is that the presence of quote marks or apostrophes is needed in an expression but not here.

Examples

```
APPUI & 1               ==> APPUI1

APPUI & AA & 1          ==> APPUIAA1

APPUI & 1 & AA & 1      ==> APPUI1AA1

n=5
APPUI & n               ==> APPUI5

n=5
("APPUI" + n)           ==> APPUI5
```

2.17 – REPEAT LOOP

Presentation

It allows repetition of a series of data or instructions.

Syntax

POUR i = list

<<

Instruction [Instruction …]

>>

POUR i = list << Instruction [Instruction ...] >>

Where:

- i: variable name,
- list: list of values assigned to the variable i in a named list, explicit lists or sequences of values separated by ",".
- Instruction: instruction or end of line.

Specifications

- A repeat loop may be nested within another loop.
- The SKIP instruction used to go directly to the end of the current loop.
- The BREAK instruction stops the iterative process.

Example 1

```
k=0
POUR i=1 a 10
   <<
    POUR j=1 a 5
       <<
        k=k+1
        SI (k>10) BRISER
        k # Value provided as data
       >>
   >>
```

Example 2

```
k=0
POUR i=1 a 10 << POUR j=1 a 5  << k=k+1 SI (k>10) BRISER k >> >>
```

Exemple 3

In the case of a PCP command needing a title and included in a repeat loop, it is interesting to parametrize this title in order to create as many different titles as there are iterations :

```
    POUR j=1 a 5
```

```
<<
name = ('bridge support subsidence' + j)
TITRE name
>>
```

This is particularly interesting to display results : tiltes are different, so PCP will find different cases and will display curves with different colors.

2.19 - SKIP INSTRUCTION

Presentation

This instruction will ignore following instructions up to the end of the current loop.

Syntax

PASSER

Specifications

- It must be placed in a POUR instruction.
- It ignores all that follows up to the first closing double angle bracket ">>" of the current loop and continues the iterative process of the current loop.

Example 1

```
k=0
Pour i = 1 a 10
   <<
   SI ( i == 5 ) PASSER
   K=k+1
   >>
```

is equivalent to

```
k=0
Pour i = 1 a 4,6 a 10
   <<
   k=k+1
   >>
```

Example 2

```
k=0
Pour i = 1 a 10
   <<
   Pour j = 1 a 10
      <<
      SI (j == 5) PASSER
      k=k+1
      >>
   >>
```

is equivalent to

```
k=0
Pour i = 1 a 10
   <<
   Pour j = 1 a 4,6 a 10
      <<
      k=k+1
      >>
   >>
```

2.19 - BREAK INSTRUCTION

Presentation

This instruction is used to stop the process of a loop and return control to the encompassing instruction.

Syntax

BRISER

Specifications

- It must be placed in a POUR instruction.
- It ignores all that follows up to the first closing double angle bracket ">>" of the current loop and stops the iterative process of the current loop.
- If there are several nested loops, it is equivalent to a "GOTO" to the instruction following the end of the loop being processed.

Example 1

```
k=0
Pour i = 1 a 10
   <<
   SI (i == 5) BRISER
   k=k+1
   >>
```

is equivalent to

```
k=0
Pour i = 1 a 4
   <<
   k=k+1
   >>
```

Example 2

```
k=0
Pour i = 1 a 10
   <<
   Pour j = 1 a 10
      <<
      SI (j == 5) BRISER
      k=k+1
      >>
   >>
```

is equivalent to

```
k=0
Pour i = 1 a 10
   <<
   Pour j = 1 a 4
      <<
      k=k+1
      >>
   >>
```

2.19 - CONDITIONAL INSTRUCTION

Presentation

It allows certain instructions to be performed or not based on tested values.

Syntax

SI (logical_expression) single instruction or instruction block

[SINON single instruction or instruction block]

Where:

- Single instruction: instruction that has no end of line.
- Instruction block: << series of single instructions or line ends >>

Specifications

- A POUR instruction written on one line is a single instruction.
- A conditional instruction written on one line is a single instruction.

Example

```
k=0
SI ( i > 2 )
   <<
    SI ( j == 3 ) POUR jj=1 a 10 << APPUI (jj) >>
    SINON
       <<
        jj=1   APPUI  jj
        jj=2   APPUI  jj
       >>
   >>
SINON  APPUI 2
```

2.21 – LOGICAL EXPRESSION

Presentation

It allows for boolean operations. Logical expressions consist of operators and combinations of operators enclosed in parentheses, that combine them with each other.

Comparison Operators

Function	symbol
Equal	= or ==
different	/=
greater than	>
less than	<
greater than or equal	>=
less than or equal	<=

Table 2.7 - Comparison operators

Logical operators

Function	symbol
Logical OR	OR
Logical AND	AND
Negation	NOT

Table 2.8 - Logical operators

Specifications

- ☐ Logical expressions obey the same combinatorial logic as Fortran expressions.
- ☐ A logical expression can only appear in a conditional instruction.

Examples

```
SI ( a1==a2 ET (a1>2 OU a2>3) )
   <<
   NŒUD 1
   >>
SINON
   <<
   NŒUD 2
   >>

SI (NON(a==1) ET NON(a==3)) NŒUD 5
```

2.22 - READ INSTRUCTION

Presentation

This instruction is used to read data from a file.

Syntax

LIRE File

LIRE 'File'

LIRE "File"

Where:

- File: name of the file to read

Specifications

- The file must be placed in the current directory.
- The case of the file name is taken into account.

Example

```
Lire 'SECTION1.DON'
```

2.22 - RETURN INSTRUCTION

Presentation

This instruction is used to stop reading data from a file.

Syntax

RETOURNER

Specifications

- The part of the data file placed after this instruction is ignored.
- It is generally used as a conditional instruction.

Example

```
SI (napp == 0) RETOURNER
```

2.24 - LANGUAGE DIFFERENCES PCP/ST1

Function	PCP language	ST1 language
Variable assignment	< x = v > x=v	x=v
Named list	l=v1,,,,vi,,,,vn LIS l=v1,,,,vi,,,,vn	l=v1,,,,vi,,,,vn
Two-dimensional array	DIM t(nl,nc)	No two-dimensional array
Global array initialization	DIM t(nl)=1 a nl DIM t(nl)=1 T=1 a nl	No global initialization
Repeater	n*v	No repeater
Advanced incrementor	I a j inc n	I a j pas n
Concatenation	('aaaa'+i) Aaaa & i	Aaaa & i
Expressions	(Parenthesized expression)	Expression without space (Parenthesized expression)
Vector expressions	Yes	No
Matrix Expressions	Yes	No
Directed List	<< expressions, ,, > v= list >	No directed List

Table 2.9 – Language differences table

Chapter 3

Basic formwork and cablings

Introduction

This chapter describes the data needed for the definition of a (spatial) beam, including its formwork, its "minimum" cabling (before any duplication) and its materials.

Data

One file should be produced per beam, its size is not limited. In particular, the number of commands to be introduced is not limited, as well as the possible data volume associated with a particular command. For example, we can introduce as many generic sections and cables as required, and as many defining points of a section or a cable as necessary (dynamically allocated memory).

"Excess" commands, and excess data in a command are not tolerated (unallocated reserve sections, surplus cable points, relative to the number announced, etc.).

The data, presented in a logical and hierarchical order for their description, can be introduced in any order (except for start and end of file commands), forward references are possible (partial section defined with respect to an entire section not yet defined, assigned section not yet defined, etc.).

The order of entry, however, determines the internal storage order of certain entities and their order of appearance in the numerical results (sections, cables, etc.).

With no end delimiter, each command in the GE1 module must begin on a new line and the first line break of the wording is to be observed. However, cutting data that follows a command header into lines, is unrestricted.

Unless otherwise indicated, all numbering is consecutive from one and all two-dimensional coordinates are provided in the generic reference frames $o_iy_iz_i$ of the sections.

Only basic material names of standard-cables and cable layouts which are "visible outside" of the beam data must differ from one beam to another.

Analysis mode

The GE1 module performs compilation type data analysis (coupled with a control), by hierarchical levels and successive scans of the entire file, from the broadest to the finest, stopping at the level where the first error occurs.

The number of possible errors that can be detected is not limited for a level, but the detection of an error of a certain level of severity, for a command, may prevent the search for other errors on the same level.

For example, meeting an incorrect keyword in a section prevents analysis of the content and the detection of a lexically incorrect integer or real value in an (alpha)numerical list will stop its analysis.

Processing sequence

If the GE1 module has detected no errors at the lowest level, the GE3 module is launched, which performs certain additional controls.

The GE5 module executes only if the GE3 module has detected no errors. If it completes successfully, the beam is recorded in a database, and can be used to be drawn or integrated into the overall mechanical model.

Editing

The GE1 module provides a reminder of the data entered in a clear form, in addition to a first state of transformation.

The GE3 and GE5 modules provide the results of the various geometrical and mechanical processing.

The volume of these outputs is globally scalable.

Contents

3.1 - BEAM

POUTRE no_poutre

Parameters

- no_poutre: identification number of the beam, from 1 to 9999. The numbering of the beams may not be consecutive *(external* numbering).

Functions

This command marks the beginning of the beam data and provides a number under which it will be saved, if validated.

If this number corresponds to a beam registered in the database, the new beam shall replace the old one.

The order of recording beams of a model is not imposed with respect to their numbers.

Conditions of use

- ☐ Must be the first line of the command file.

Examples

```
POUTRE  100
...........
...........
FIN
```

Related commands

TITRE ; MATERIAU ; CARACTERISTIQUES CABLES ; TRACE CABLE ; FIN

3.2 - TITLE

TITRE   titre_poutre

Parameters

- titre_poutre: character string.

Functions

The first TITRE command met holds the main title of the beam, which is at the top of each page of the digital outputs (of the GE1, GE2, GE3, GE4, GE5 or MC2 modules), and each drawing (of the GE2 or GE4 modules), thereof.

This main title also appears in the results of the PH1 module.

The content of all TITRE commands (including the first) is also reproduced at the beginning of the results of the GE1 module.

Conditions of use

☐ Introduce at least one command of this type.

Examples

```
TITRE  '*** VIADUC D''ACCES B, TABLIER ***'
TITRE  '--------------------------------'
```

Related commands

POUTRE

3.3 - GENERAL

GENERALITES nb_sigma nb_tau type_s orig_s type_sp type_gth nb_hou type_noe

Parameters

Refer to Figure 3.4 and the TRACE CABLE command.

For all sections, constant and positive numbers of constraint calculation points of each of the two types below are assumed, referred to as "calculation fibers".

- nb_sigma: number of normal stress calculation fibers, positive;
- nb_tau: number of tangential and normal stress calculation fibers, positive;
- type_s: calculation mode of approximating curvilinear abscissa of points of the reference fiber; these abscissa optionally serve to position some of the cables along the beam; the abscissa of cable definition points/poles that use them must be defined according to the same convention, indicate:
  - 1 if the reference fiber is projected onto the $X_bO_bY_b$ plane of the beam reference frame (top view);
  - 2 if we are working in three dimensions;
- orig_s: position of the curvilinear abscissa origin, relative to the first section, supposed common to all the longitudinally positioned cables in the absolute reference frame; this value is also used by the MC1 module to operate absolute longitudinal type cabling transformations;
- type_sp: means of getting "surface/outer perimeter" reports for sections; these values are used by the PH3 module, in CALCUL RHEOLOGIQUE FIN mode, in certain materials shrinkage and creep laws (see reference documents listed in Annex B), indicate:
  - 1 if these values are calculated, with a coating width provided for each section to be deducted eventually from the outer perimeter;
  - 2 if these values are given;
- type_gth: means of getting generic envelope coordinates of all sections, needed for the PH3 module to assess the effects of thermal gradients applied directly to the beam (see Chapter 8, options LOCAL Z and POUTRE of the CHARGEMENT [IDENTIQUE] THERMIQUE command), indicate:
  - 1 if these coordinates are calculated for all points of each section (true envelope values);
  - 2 if these coordinates must be calculated along a parallel to the generic axis o_iz_i passing through the center of gravity of each section (encountered material envelope);
- nb_hou: number of beam slabs, indicate:
  - 0 if the beam is not constituted exclusively of sections with contoured morphology, or if it is not desired to calculate the coefficients correcting tangential stresses by bending forces (Résal effect due to the normal forces and bending moments);
  - a positive value for the number of "generalized" section slabs, formed by excluding the "webs" and their extensions (Figure 3.1), and assumed constant along the beam otherwise.

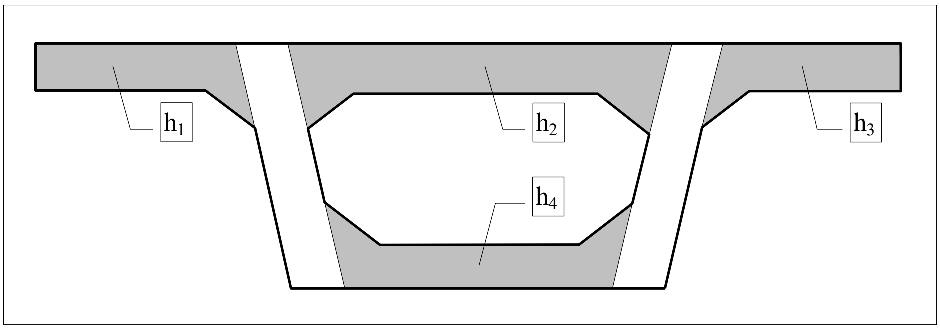


Figure 3.1 - Generalized 4 slab section

- type_noe: processing mode of contoured section node mean lines, indicate:
  - 0 if there are no such sections
  - 1 if the centers of gravity of the "octopus-like" modeling elements must coincide with those of the nodes;
  - 2 if the centers of gravity of the nodes must be taken as center points of these octopus-like elements.

Functions

This command provides general information and options, applying to the entire beam.

Conditions of use

- Introduce only one command of this type.

Methodological advice

- If the reference fiber is contained in a plane parallel to plane $X_bO_bY_b$of the beam reference frame, the approximating curvilinear abscissa can be calculated in two or three dimensions (equivalent results), if this is not the case, we generally work in three dimensions.
- The choice of method for calculating the envelope coordinates for thermal gradients is directly related to the shape of the section.

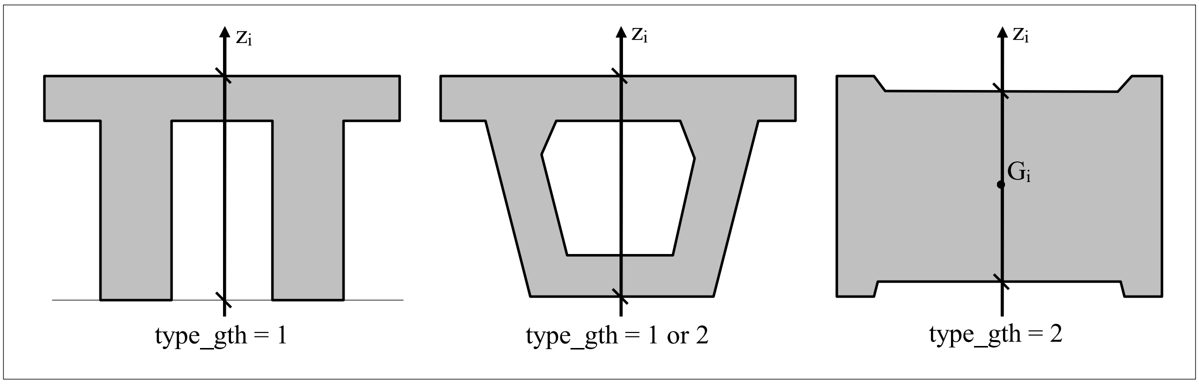


Figure 3.2 - Formwork envelopes for thermal gradients

- The choice of processing mode for contoured section nodes influences their mechanical shear characteristics and calculated tangential stresses.
- With type_noe = 1, the static moments of the octopus-like element modeling a node exactly balance those of the corresponding polygon. However, the breakdown is more difficult to achieve, the shear flow (mean) lines are often elongated, the wall thickness is reduced, and the sections are more flexible, with the shear stresses being modified.
- In most cases, the center of gravity shifts due to a node are completely compensated (in the case of symmetry), or partially, and the choice type_noe = 2 simplifies the breakdown and improves the quality of the mechanical characteristics in question and the shear flows.
- As a general rule, choose type_noe = 1 if we want to maintain strict compatibility with "previous"calculation results.
- Otherwise, the choice type_noe = 2 will give better results, especially in the absence of gussets at node branch junctions and the "lean" node breakdown.

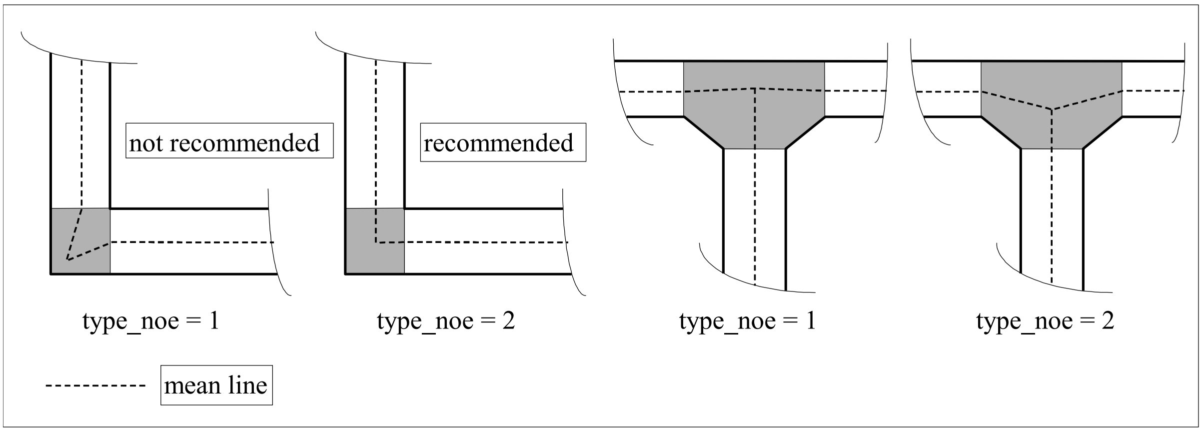


Figure 3.3 - Contoured sections, mean node lines according to processing mode

Examples

```
$            n°_sigma n°_tau type_s orig_s type_sp type_gth nb_hou type_node
GENERALITES  2        4      2      0.0    1       1        4      1
```

Related commands

FIBRE REPERE ; SECTION MASSIVE ; SECTION CONTOURS ENTIERE

SECTION CONTOURS PARTIELLE ; SECTION PAROIS ; EPAISSEURS PAROIS

TRACE CABLE

3.4 - REFERENCE FIBER

FIBRE REPERE nb_points nb_angles

$$\left\langle \text{x\_point} \quad \text{y\_point} \quad \text{z\_point} \quad \langle \theta_i \rangle_{\text{nb\_angles}} \right\rangle_{\text{nb\_points}}$$

Parameters

- nb_points: number of reference fiber definition points, at least 2; the numbering of its points or sections determines the direction of travel along the beam;
- nb_angles: number of generic reference frame positioning angles, always equal to 3;
- x_point, y_point, z_point: the origin of coordinates of a generic reference frame, in the beam reference frame; between consecutive points maintain a distance of 0.02 m (calculated according to the value assigned to the parameter type_s of the GENERALITES command);
- θ_1: plane alignment angle, keeping the same definition for all points (abrupt variations forbidden);
- θ_2: lengthwise profile angle, absolute value less than 85.0;
- θ_3: tilt angle, absolute value less than 45.0.

This triple normalized rotation (see Figure 1.2) orients the current generic reference frame relative to the beam reference frame.

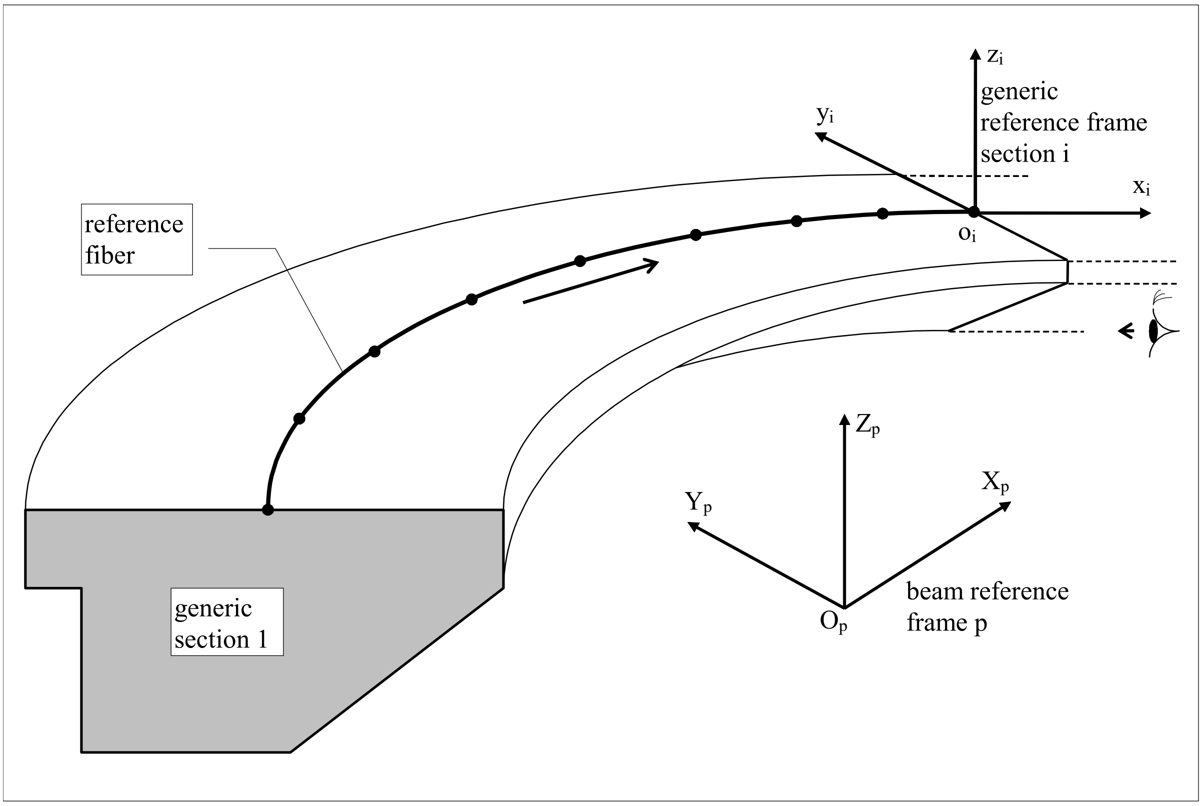


Figure 3.4 - Reference fiber

Functions

This command defines the reference line point by point (reference fiber) in the beam reference frame and the longitudinal breakdown of the beam in defined sections (generic sections).

It should be said that the generic section i is defined in the $o_iy_iz_i$; reference frame; the axis x_i, normal to the generic section plane, is close in orientation to that of the tangent to the reference fiber.

If we perform a cut at the section i, keeping only the beam segment [1 .. i]; the observer in the remaining part sees the section to be defined "from the right".

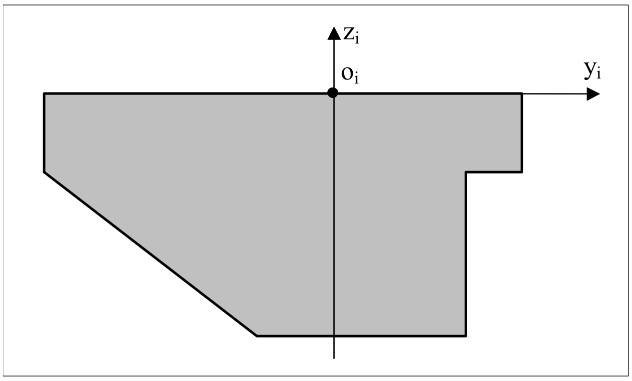


Figure 3.5 - Generic section i, view from the right, 2D reference frame

Conditions of use

- Introduce only one command of this type.

Methodological advice

- The accuracy of acquisition of coordinates is best when the origin of the reference frame attached to a beam is close to the "middle" of its reference fiber; this recommendation should particularly be applied to curved beams whose length exceeds 1000.0.
- The X_p axis of the beam reference frame may be oriented substantially in the "mean direction of beam development"; its direction thus often coincides with the direction of travel chosen (not mandatory).
- If the beam sections have a variable part and a constant part, we can attach the origin of the current generic reference frame to the constant part of each section (slabs of a caisson deck of constant width).
- If the beam actually supports traffic, the positioning of the reference fiber at the center of the road axis makes it possible to be reused as a load SUPPORT element (see Section 9).
- If the sections have a formwork and/or cabling axis of symmetry, we can position the o_iz_i axis on that axis or parallel to it.
- The cable crossing points being defined in generic reference frames, it is necessary to position the o_iy_i axis so as to enter the coordinates in the most convenient way.

- For the plane inclination of the generic reference frame, we may choose the reference fiber tangent.
- For inclination in elevation, the choice is determined by the mode of definition of the cable points/poles outside the sections, between the reference fiber tangent or a nearby direction (vertical or non-vertical generic sections on a sloping deck).
- The tilt angle can be integrated in the section at its definition if it is not transversely symmetrical.
- A modeling and calculation section corresponds to each generic section, and at least one node of the general mechanical model.
- Integrate the geometric features of the formwork, in particular, absorb section discontinuities with "short" transition elements, which will absorb the mean fiber jumps (see TRONCONS command).
- Tighten up the sections in parts of beams where the characteristics vary rapidly.
- Take into account all sections actually used to simulate the construction process (segment joints, etc.).
- The longitudinal breakdown of a beam pushed in place should verify particular constraints (see Annex D).
- Integrate all sections receiving provisional or definitive supports.
- If using the MC1 module for generating cables by the relative type of longitudinal transformations, the parts of beams where the basic cables are defined and those receiving duplicated cables must have the same distribution of sections.
- Basic material changes take place in generic section planes.
- Regulatory considerations may lead to the addition of verification sections.
- When applicable, the shear corrections due to the Résal effect are assumed to develop, in each section, in a direction parallel to the o_iz_i axis.

Examples

The reference fiber below is carried by a cylinder with an axis parallel to O_bZ_b, its projection on $X_bO_bZ_b$ is rectilinear, with a slope of 5%; it is an elliptical arc in space.

The angle of tilt, assumed constant, corresponds to a slope of 2.5%.

The origin of the beam reference frame is placed at the first point of the reference fiber and its axis X_p is on the chord of the arc of the circle viewed in plan.

Notice the positioning angle signs of generic frame i:

- θ_1 is counted positive, following O_bZ_b, from O_bX_b towards O_bY_b;
- θ_2 is counted positive, following o_iy_i, from o_iz_i towards o_ix_i ;
- θ_3 is counted positive, following o_ix_i, from o_iy_i towards o_iz_i.

The breakdown is done in 96 segments (97 points), according to a constant angular pitch of 1.25 degrees.

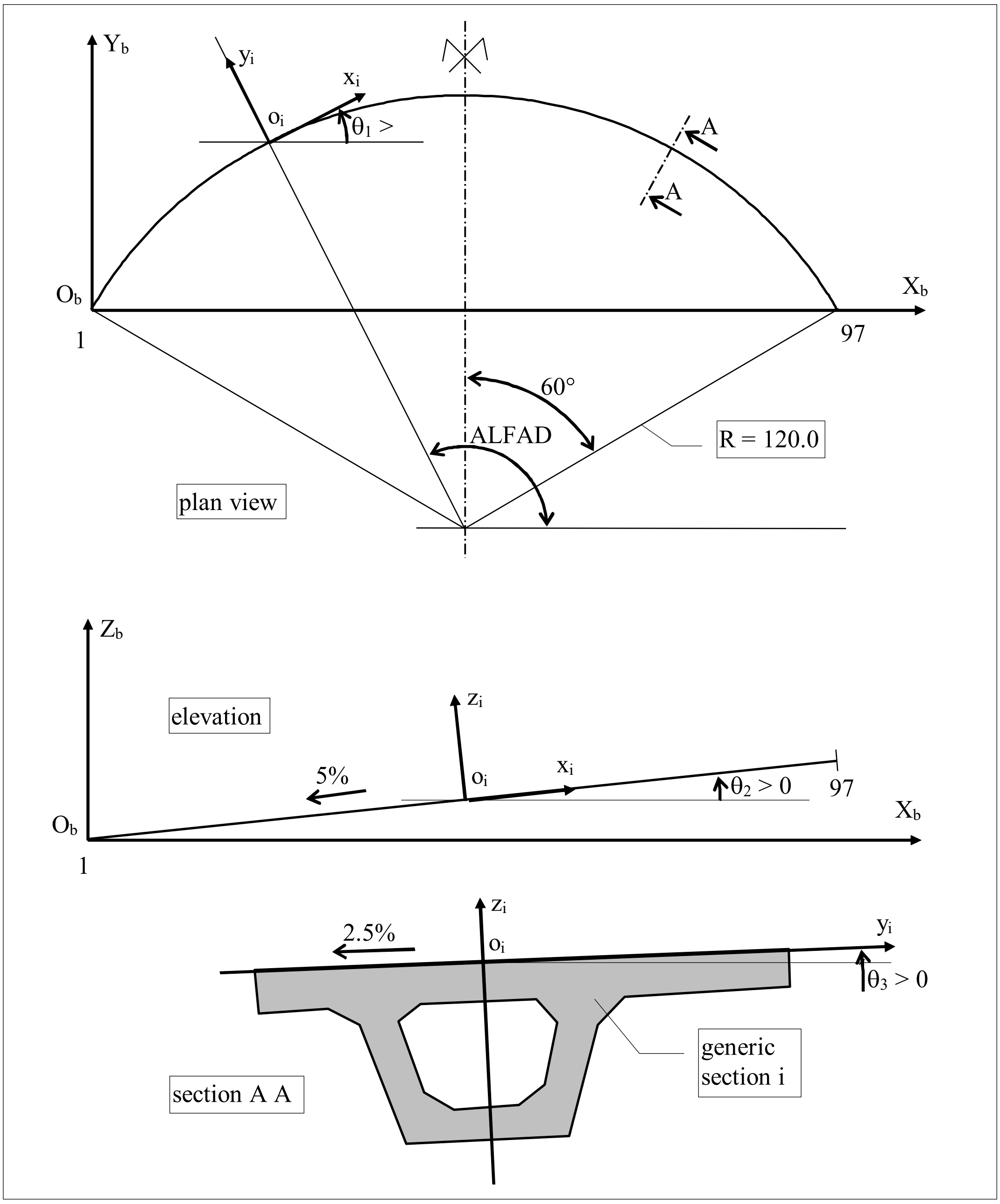


Figure 3.6 - Elliptical reference fiber

```
<R = 120.0>
FIBRE REPERE  97  3
<<<<
     (X, R*SIN(ALFAR)-60.0, 0.05*X, ALFAD-90.0, -2.86241, 1.4321)>
     X     = (R*COS(ALFAR)+103.92305)>
     ALFAR = (0.0174533*ALFAD)>
     ALFAD = 150.0 A 30.0 DIV 96>
```

Related commands

GENERALITES ; AFFECTATION SECTION ; TRONCONS ; ARTICULATIONS

EPAISSEURS PAROIS ; MATERIAU ; AFFECTATION MATERIAUX

TRACE CABLE

3.5 - MASSIVE SECTION

SECTION MASSIVE nom_section nb_evid

$\text{nb\_points\_ext} \quad [\langle \text{nb\_points\_evid} \rangle_{\text{nb\_evid}}]$

$\langle \text{y\_point} \quad \text{z\_point} \rangle_{\text{nb\_points\_ext}} \quad [\langle\langle \text{y\_point} \quad \text{z\_point} \rangle_{\text{nb\_points\_evid}} \rangle_{\text{nb\_evid}}]$

$\langle \text{y\_sigma} \quad \text{z\_sigma} \rangle_{\text{nb\_sigma}} \quad \langle \text{y\_tau} \quad \text{z\_tau} \rangle_{\text{nb\_tau}}$

rigid_fy rigid_fz inertie_mx yp_centre zp_centre

$\langle \text{flux\_fy} \quad \text{flux\_fz} \quad \text{flux\_mx} \rangle_{\text{nb\_tau}} \quad \text{larg\_revet}$

Parameters

The nb_sigma, nb_tau and type_sp parameters are provided by the GENERALITES command; refer to Figure 3.7.

- nom_section: section name, which cannot be assigned to another cable section;
- nb_evid: number of cavities, positive if the section has cavities, zero otherwise;
- nb_points_ext: number of points of the outer contour, at least 3;
- nb_points_evid: number of points of a cavity, at least 3;
  - y_point, z_point: coordinates of a definition point of the section; the cavities are introduced in the same order as their number of points; each polygon (open) is described by turning counterclockwise (from o_iy_i towards o_iz_i), the first point is freely chosen;
- y_sigma, z_sigma: coordinates of a point for the calculation of normal stresses;
- y_tau, z_tau: coordinates of a point for the calculation of tangential and normal stresses; these coordinates are only used to calculate the normal stresses (concomitant);
- rigid_fy, rigid_fz: shear stiffness depending on the principal axes G_iyp_i and G_izp_i, positive or zero; in the absence of a precise knowledge of these parameters, we can introduce:
  - shear force reduced sections;
  - zero values indicating that the deformation due to shear force is not taken into account;
- inertie_mx: pure torsion inertia (St Venant constant), positive or zero;
- yp_centre, zp_centre: coordinates of the center of torsion in the *principal reference frame;*
- flux_fy, flux_fz: shear flow generated by the unit shear forces applied by G_iyp_i and G_izp_i; the tangential stresses (mean) are still obtained by dividing the flow values by the corresponding wall thicknesses (see EPAISSEURS PAROIS command);
- flux_mx: shear flow produced by a unit torsion moment;
- larg_revet: coating width (possibly zero), to be deducted from the outer perimeter of the section before calculating the surface/outside perimeter ratio if type_sp is 1; value provided of the surface/outer perimeter of the section if type_sp is 2.

Functions

This command provides a section that is or is taken to be massive, always defined in full.

Conditions of use

- The use of this command is optional.

Methodological advice

- If we are not interested in the tangential stress values, shear flow values can be entered as zero unit forces.
- The valuesof unit flows flux_fy and flux_fz should be consistent with the net thicknesses provided in the EPAISSEURS PAROIS command; these are fictitious thicknesses that can be made equal to 1.0, if we choose to do so or if the section is purely massive; flows are then identified with stresses.

Examples

The section with two cavities below, described as a MASSIVE SECTION, is enclosed in a rectangle of 7.20 m x 3 m.

We find this section, described as a contoured section with morphology, in the SECTION CONTOURS ENTIERE command example. Compare both definitions.

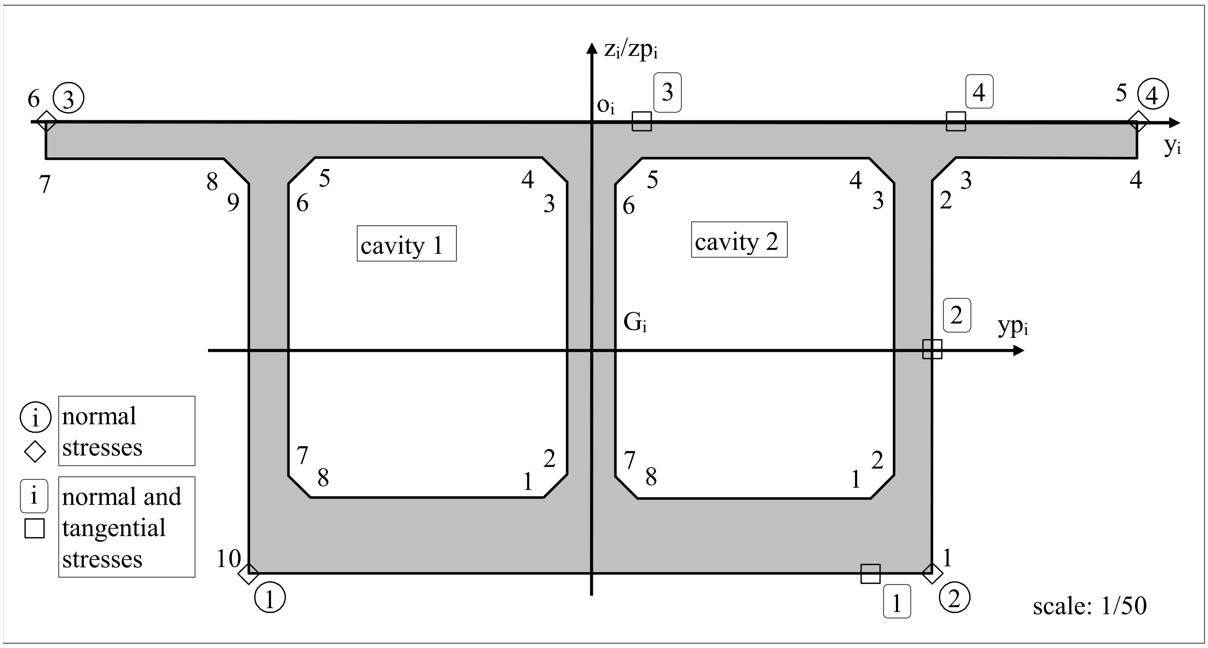


Figure 3.7 - Section with cavity considered "massive"

```
$           n°_sigma n°_tau type_s orig_s type_sp type_gth nb_hou type_node
GENERALITES  4        4      1      0.0    1       1        0      2
..........................
SECTION MASSIVE  CAISSON1  2
$ number of points of the contours
10 8 8
$ exterior contour
 2.25 -3.00   2.25 -0.40   2.40 -0.25   3.60 -0.25
 3.60  0.00  -3.60  0.00  -3.60 -0.25  -2.40 -0.25
-2.25 -0.40  -2.25 -3.00
$ cutting 1
-0.30 -2.50  -0.15 -2.35  -0.15 -0.40  -0.30 -0.25
-1.85 -0.25  -2.00 -0.40  -2.00 -2.35  -1.85 -2.50
$ cutting 2
 1.85 -2.50   2.00 -2.35   2.00 -0.40   1.85 -0.25
 0.30 -0.25   0.15 -0.40   0.15 -2.35   0.30 -2.50
$ calculation points of the normal stresses
-2.25 -3.00   2.25 -3.00  -3.60  0.00   3.60  0.00
$ calculation points of the tangential and normal stresses
 1.85 -3.00   2.25 -1.495  0.30  0.00   2.40  0.00
$ mechanical shear-torsion characteristics
$ rigid_fy  rigid_fz  inertiea_mx  yp_centre  zp_centre
  3.001     1.865     10.783      0.0000     -0.2180
$ shear flows under unit loads
$ entered in the same order as the calculation points
$ of the tangential and normal stresses
$ flow fy  flow_fz    flow_mx
  0.08696  0.08044    0.04374
  0.03048  0.12889    0.04374
 -0.10693 -0.04594    0.04374
  0.05314  0.05307    0.00000
$ width of the coating (calculated surface area/perimeter ratios)
7.20
```

Related commands

GENERALITES ; AFFECTATION SECTION ; EPAISSEURS PAROIS

3.6 - ENTIRE SECTION CONTOURS

SECTION CONTOURS ENTIERE nom_section nb_evid

$nb\_points\_ext \ [\langle nb\_points\_evid \rangle_{nb\_evid}]$

$[\langle nb\_points\_hou \rangle_{nb\_hou}]$

$\langle y\_point \quad z\_point \quad ordre\_noeud \quad noeud \rangle_{nb\_points\_ext}$

$[\langle\langle y\_point \quad z\_point \quad ordre\_noeud \quad noeud \rangle_{nb\_points\_evid} \rangle_{nb\_evid}]$

$[\langle\langle ordre\_contour \quad contour \rangle_{nb\_points\_hou} \rangle_{nb\_hou}]$

$\langle y\_sigma \quad z\_sigma \rangle_{nb\_sigma} \quad \langle y\_tau \quad z\_tau \rangle_{nb\_tau} \quad larg\_revet$

Parameters

The nb_sigma nb_tau, type_sp and nb_hou parameters are provided by the GENERALITES command.

- nom_section: section name, which cannot be assigned to another beam section;
- nb_evid: number of cavities, positive if the section has cavities, zero otherwise;
- nb_points_ext: number of points of the outer contour, at least 3;
- nb_points_evid: number of points of a cavity, at least 3;
- nb_points_hou: number of points of a generalized slab, at least 3. The number of generalized slab nb_hou has been defined in the GENERALITES command. As a reminder, the generalized slabs are formed by excluding the "webs" and their extensions (Figure 3.8);

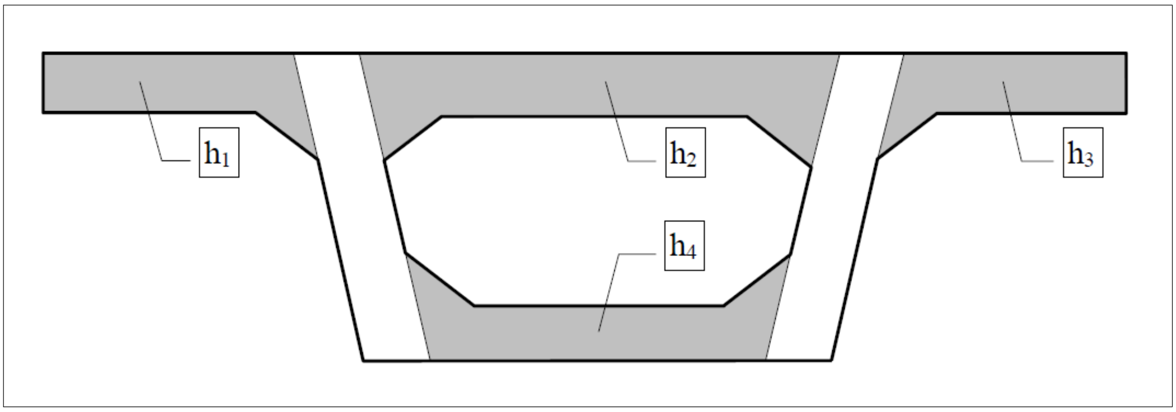


Figure 3.8 - Generalized 4 slab section

- y_point, z_point: coordinates of a definition point of the section; the cavities are introduced in the same order as their number of points; each polygon (open) is described by turning counterclockwise (from o_iy_i towards o_iz_i), the first point is freely chosen.

To calculate the shear flow at any point of the section, the GE3 module must extract a mean line called "equivalent thin-walled profile".

For this operation to be successful regardless of the shape of the section, it is necessary to define the "morphology", separating the massive parts or "nodes" from the remaining parts or "branches" by rectilinear cuts.

The walls resulting naturally from the branches, determined by transfer of bissectors, will be artificially connected by walls resulting from the nodes, to ensure the connectivity of the model and best retain the characteristics of the initial section (area, center of gravity, bending inertia)

We distinguish the branches connecting two nodes, and the "dangling" branches, with a free end that has to be detected. During the definition of the section and after having defined a node, the first end point defined in a dangling branch must be detected.

The implementation of node cutting may impose the introduction on the contours of additional points not strictly necessary for their definition.

Each node (open and having at least 3 points), identified by a number, is described counterclockwise (from o_iy_i towards o_iz_i), the first point is freely chosen.

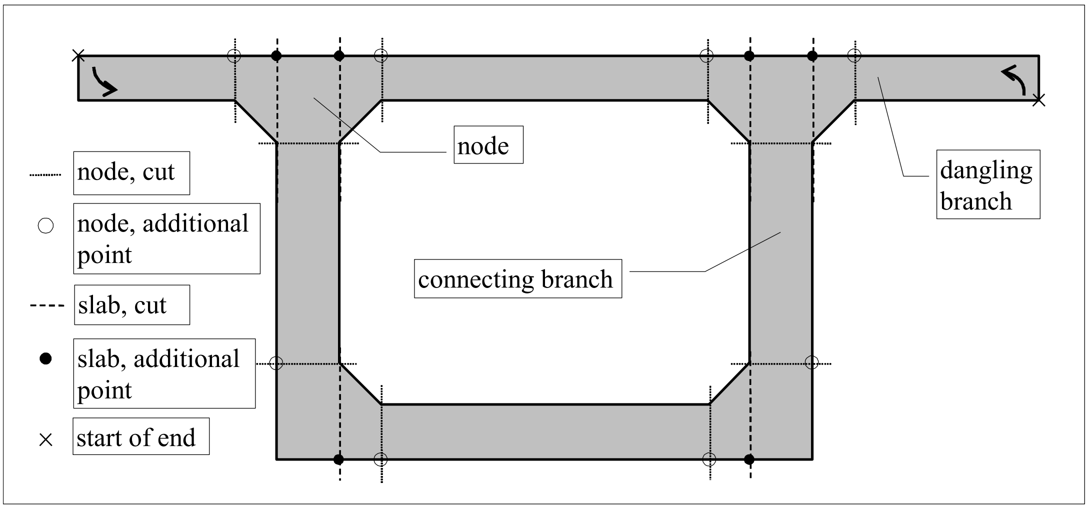


Figure 3.9 - Contoured section with morphology and slabs

- ordre_noeud: if the point belongs to a node, sequence number of the point on the node, 1 if the point is a start of a dangling branch end, 0 in other cases;
- node: if the point belongs to a node, node number, 0 otherwise.

The slabs are introduced in the same order as their numbers of points, each is to occupy the same "position", for all sections of the beam. Their inclusion may impose the introduction of extra points on the outer contour, in the extension of the main segments delimiting the webs (Figure 3.9).

Each slab (open) is described by turning counterclockwise (from o_iy_i towards o_iz_i), the first point is freely chosen.

- ordre_contour: contour sequence number of a point belonging to a slab;
- contour: corresponding contour number, 0 for the outer contour, i for the cavity i;
- y_sigma, z_sigma: coordinates of a point for the calculation of normal stresses;
- y_tau, z_tau: coordinates of a point for the calculation of tangential and normal stresses;
  - if these coordinates are both less than 1000.0, they are retained for the calculation of normal stresses; tangential stress calculation occurs on the projection of the point provided, on the wall of the closest mean line (Figure 3.10 a);

- if y_tau is less than 1000.0, and z_tau greater than or equal to 1000.0, the tangential and normal stresses calculation point is assumed to be at the intersection of the mean line with the axis Gyp_i, whose y coordinate is the closest to y_tau; in this case, y_tau may have a simple indicative value since it will be recalculated (Figure 3.10 b);
- similarly, we denote an intersection of the mean line with the Gzp_i axis, introducing a y_tau value greater than or equal to 1000.0, coupled with an indicative value of z_tau (Figure 3.10 c).

The coordinates y_tau and z_tau cannot both be greater than or equal to 1000.0.

- larg_revet: coating width (possibly zero), to be deducted from the outer perimeter of the section before calculating the surface/outside perimeter ratio if type_sp is 1; value provided of the surface/outer perimeter of the section if type_sp is 2.

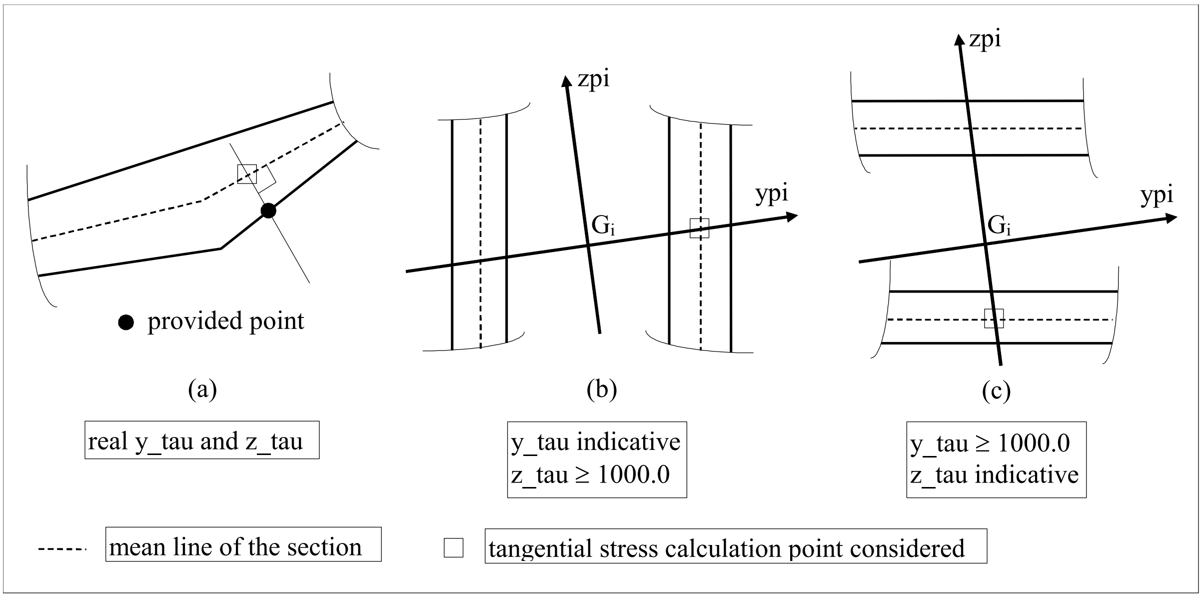


Figure 3.10 - Section with contours/walls, tangential and normal stresses

Functions

This command provides a thin-walled section entirely defined, by polygonal contours, with morphology.

Conditions of use

- The use of this command is optional.
- Use only contoured sections with morphology and slabs (full or partial) if we wish to automatically calculate tangential stress corrections due to the Résal effect.
- The application of the theory of thin-walled profiles assumes the section is transversely non-deformable.

Methodological advice

The quality of the equivalent thin profile and its calculated mechanical characteristics under tangential loading depends significantly on the placement of the nodes and their size (adjustment of cuts).

Placement of nodes

- A non-dangling branch must always connect two different nodes.
- Two given nodes cannot be connected by more than one branch.
- The minimum number of nodes to be introduced is one if the section has no cavity (minimum section formed of a node and one or more dangling branches), and three otherwise.
- A section must have at least one non-vertical branch.

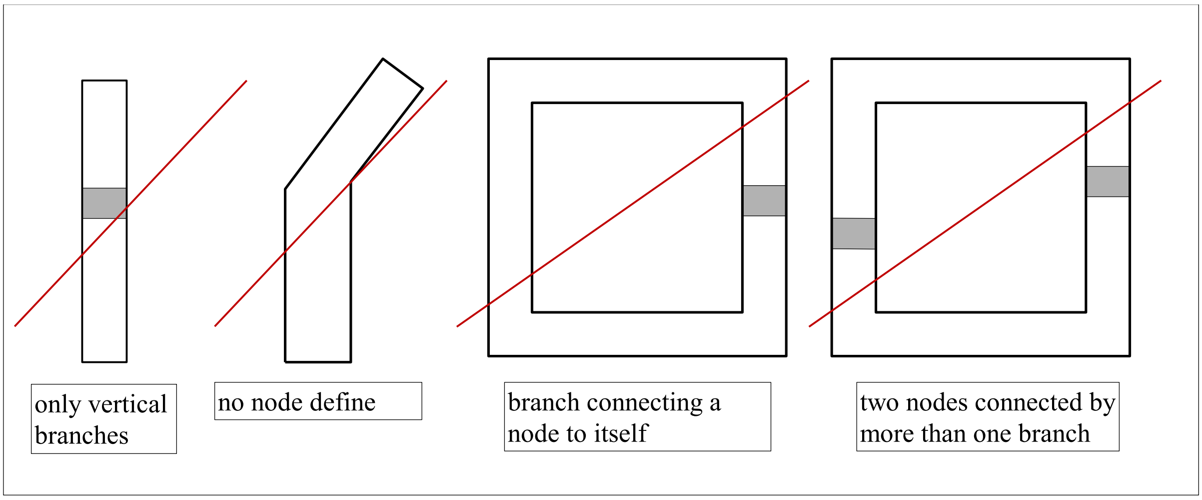


Figure 3.11 - Morphology, forbidden layouts

- The position of a node is imposed only if the corresponding massive part connects more than two branches.

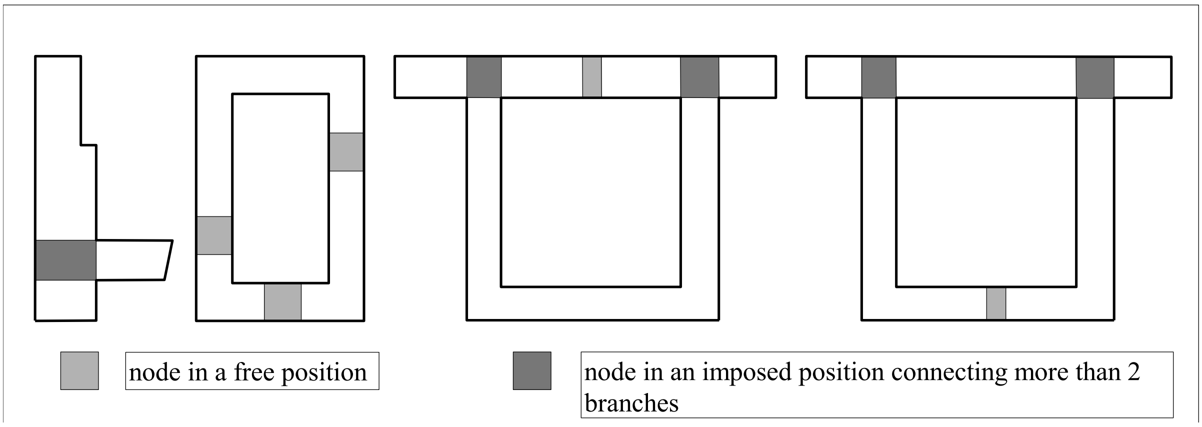


Figure 3.12 - Morphology, authorized layouts

- Where a branch introduces a sudden change of direction, the introduction of a node, at the change of direction, is required only if the mean line is incalculable.

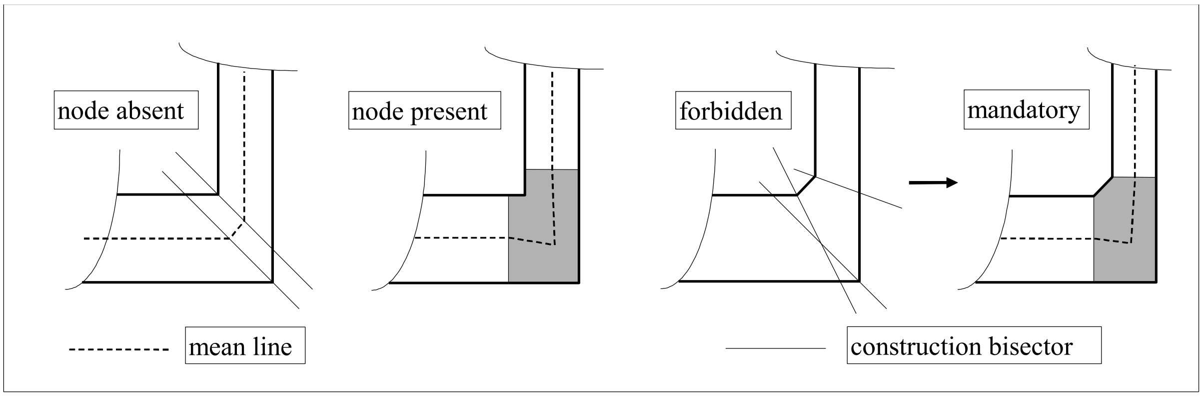


Figure 3.13 - Branch, sudden change of direction

- Where a branch possesses a geometric "accident", the introduction of a node is required only if the mean line is incalculable.

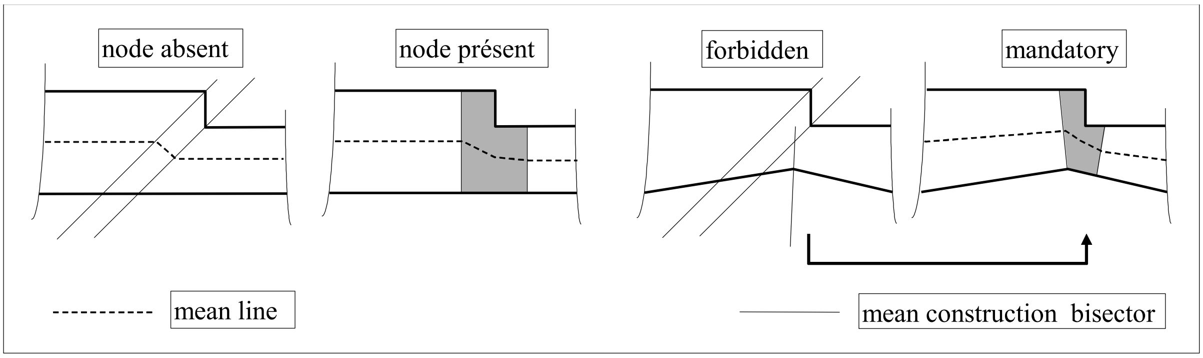


Figure 3.14 - Branch, geometric accident

Size of nodes

- Gussets of "reasonable" size can be integrated with nodes (Figure 3.15, a and b).
- Certain overly skewed cuts can prevent the calculation of the average line, due to collision of the construction bisectors with the node cuts (Figure 3.15, c).

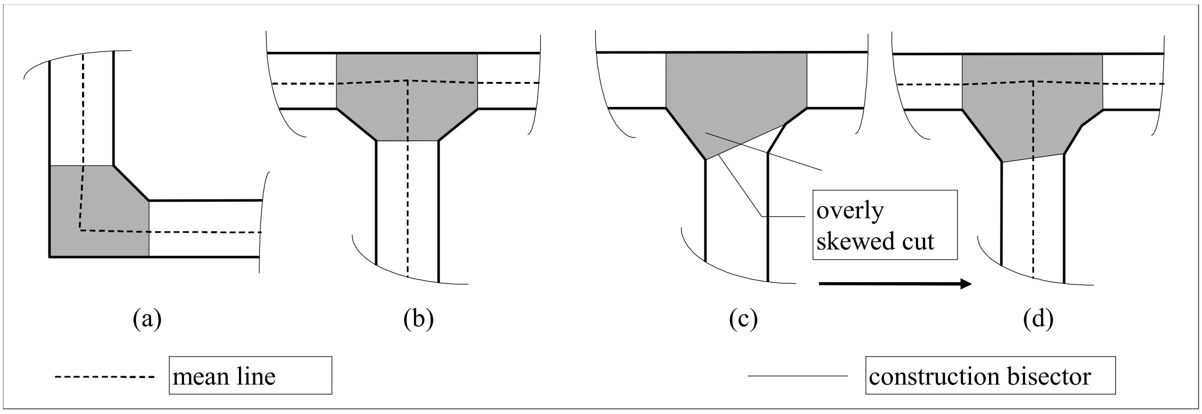


Figure 3.15 - Junctions with gussets

- On a "right angle" junction without gusset, expand the node if type_noe = 1 (see GENERALITES command), to limit the offset of the mean line (conservation of static moments); when type_noe = 2, cutting "flush" with the junction is appropriate.

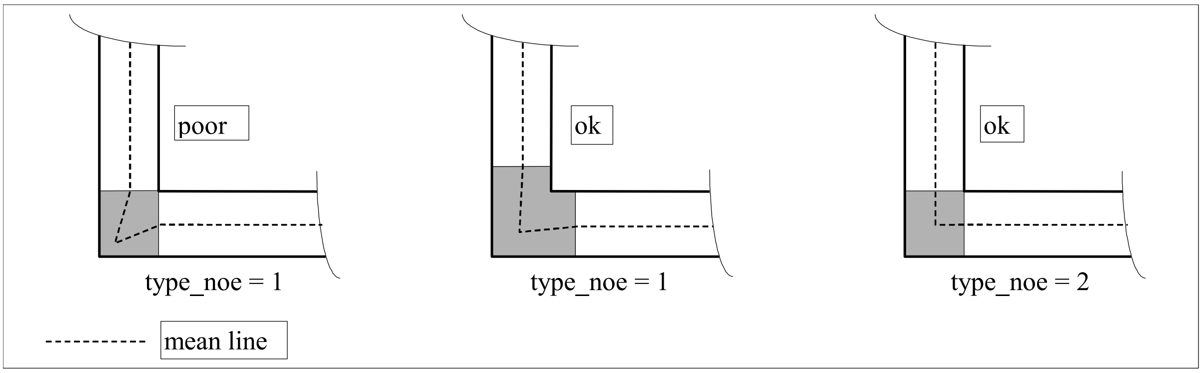


Figure 3.16 - Right angle junctions, without gusset

- It is the same on a "T" junction without gusset.

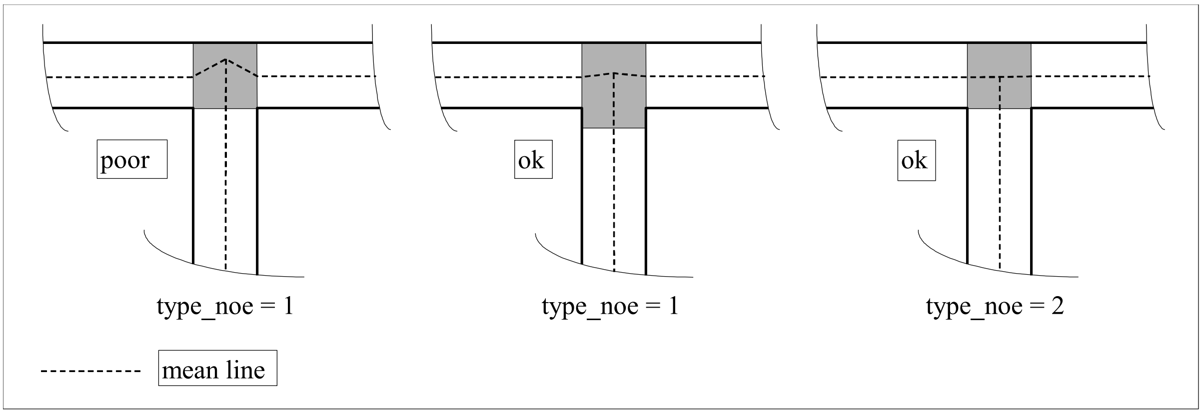


Figure 3.17 - T junctions, without gusset

Comparative study on a simple case

The section processed below, according to 5 modeling cases, has a square outer contour with 1 m sides and a rectangular cavity of 0.8 m x 0.7 m, arranged to retain an axis of symmetry. The wall thicknesses are 3 x 0.1 m and 0.2 m.

The "reference" valuesare provided by the integral equation method with an accuracy better than 0.1% (900 tabulation points on contours).

We compare, with each other and with the reference values, the mechanical characteristics of interest calculated by the GE5 module: shear rigidities according to Gyp_i and Gzp_i, St Venant constant and vertical offset of the center of torsion relative to the center of gravity.

Case 1 applies to the description of the section using the DIVISION PAROIS command (simplest direct definition of the mean line and wall thicknesses).

Cases 2-5 apply to the description of the section using the SECTION CONTOURS (ENTIERE or PARTIELLE) command; we will vary the size of the nodes. All these cases are processed with type_noe = 1 except case 3 (b) for which type_noe = 2 is chosen.

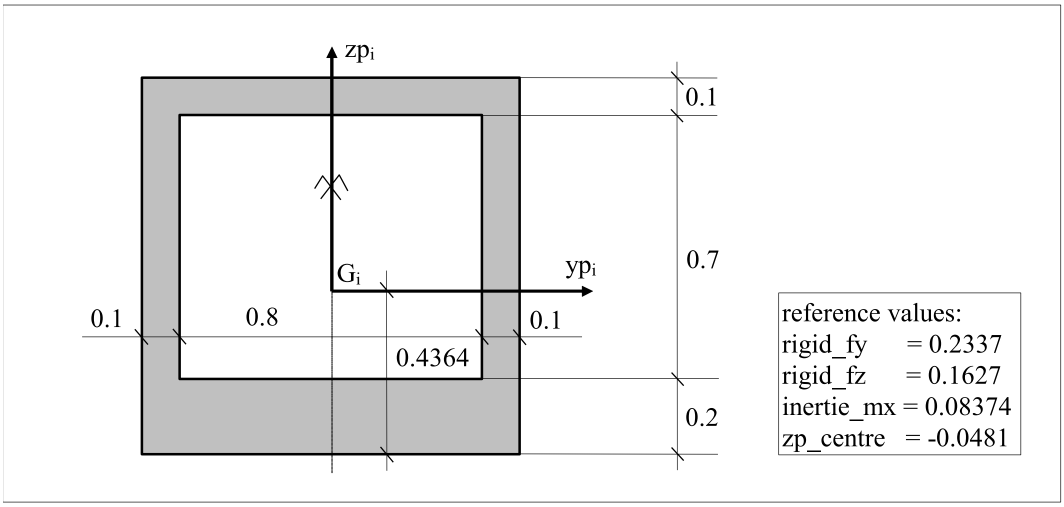


Figure 3.18 - Section for study

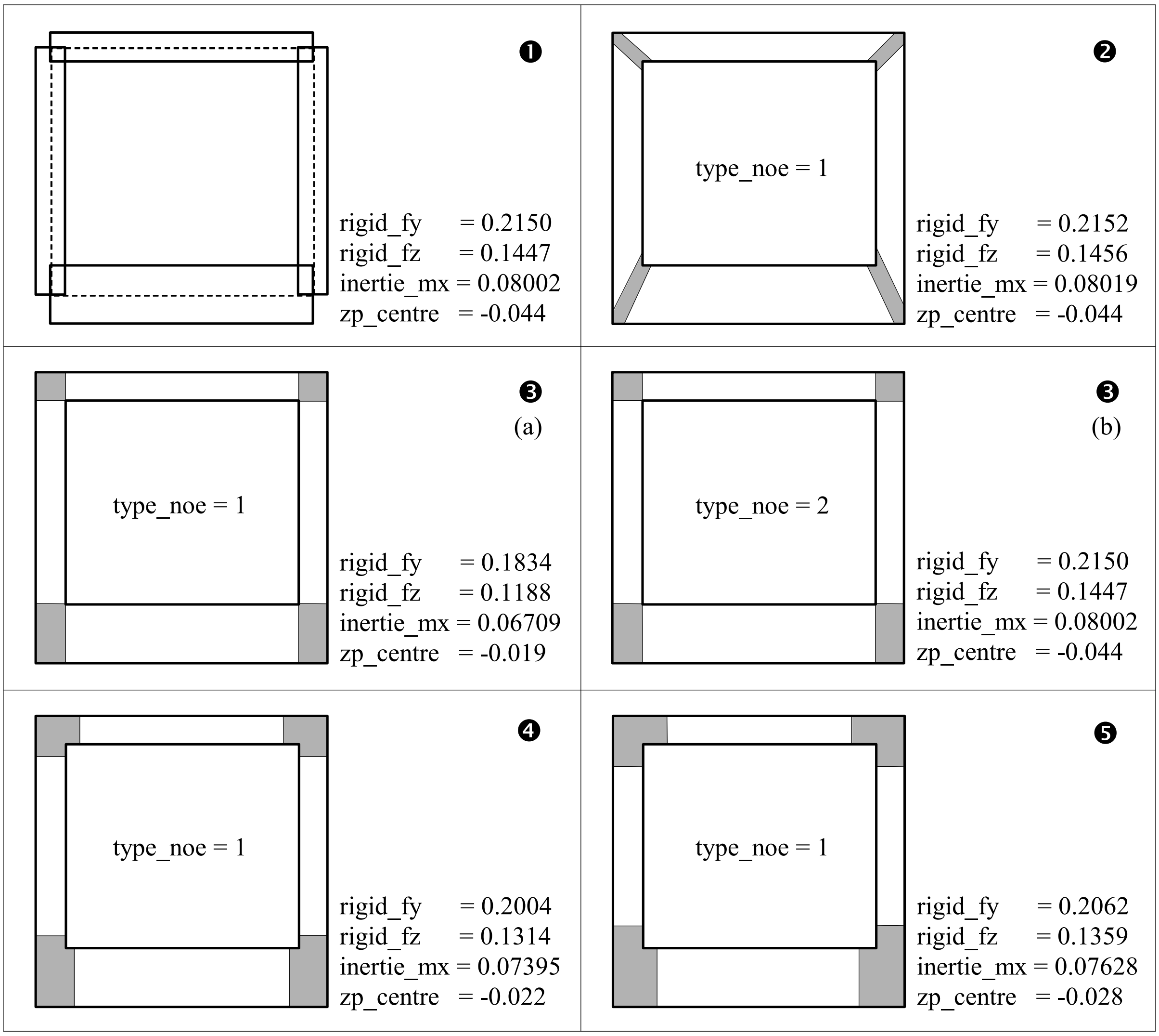


Figure 3.19 - Modeling case, values produced by the GE5 module

There are significant variations in the shear stiffness and St Venant constant, which remain below the reference values.

The center of torsion, very sensitive to modeling, remains above its reference position.

Case 2 provides good values, tending towards those of case 1 when the node surface decreases, which seems to prove, at least in this example, that we can take advantage of a "good" reduction of the nodes concurrent with the expansion we are used to (with type_noe = 1).

Case 3 (b) provides identical valuesto those of case 1, which shows the superiority of choice type_noe = 2 when the junctions are without gussets and when the inaccuracies due to this choice strongly compensate one another.

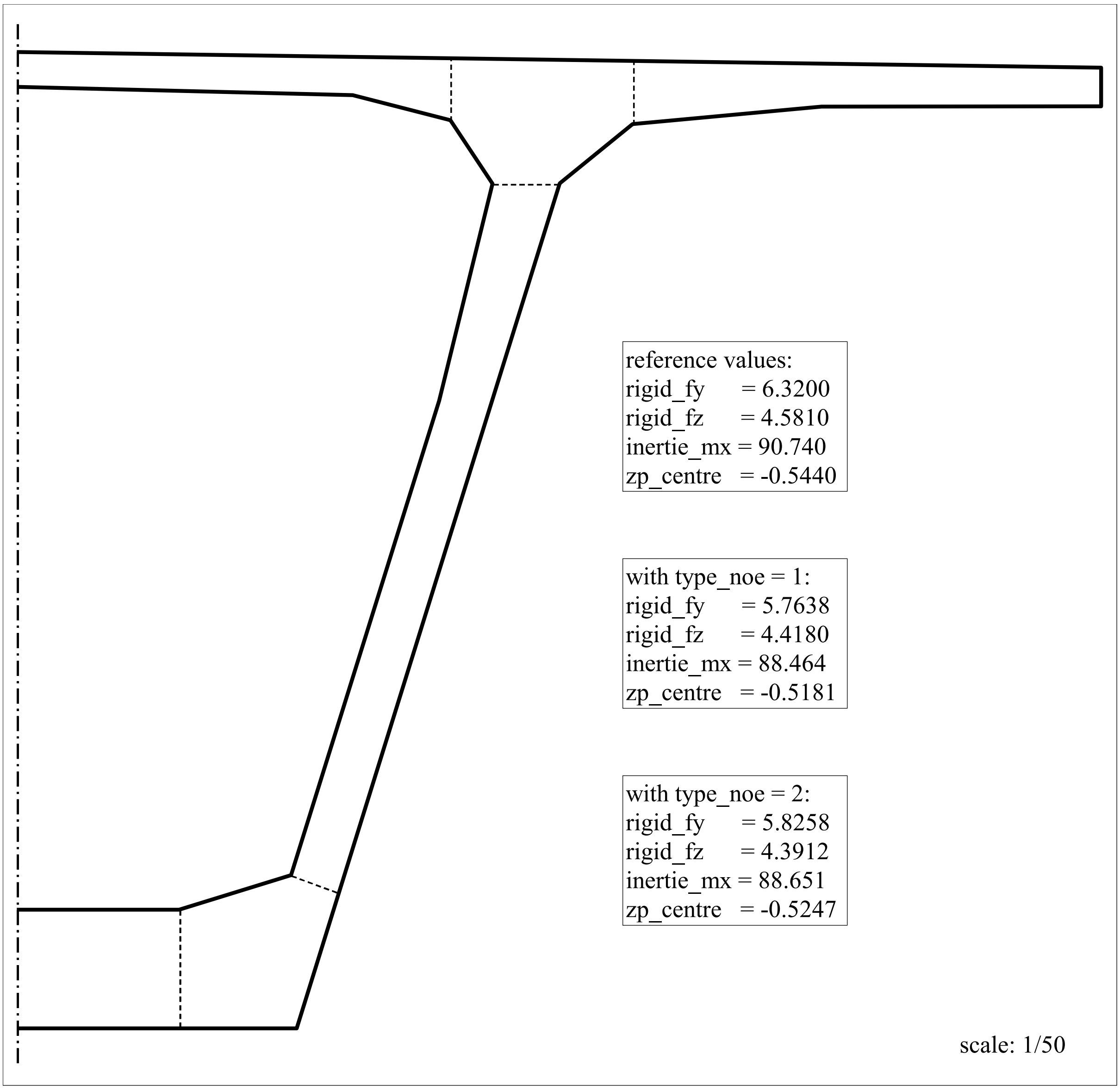
Comparative study on a real case

The transversely symmetrical section processed below (section of deck on a pier of the Ile de Ré bridge) is enclosed in a rectangle of 15.50 m x 7.00 m.

It is processed with a single node cutting (conventionally flush with gussets), taking type_noe = 1 then type_noe = 2.

The reference values are also provided by the integral equation method (with 464 tabulation points on contours).

We compare, with each other and with the reference values, the same mechanical characteristics as for the simple case processed above.



reference values:
rigid_fy = 6.3200
rigid_fz = 4.5810
inertie_mx = 90.740
zp_centre = -0.5440

with type_noe = 1:
rigid_fy = 5.7638
rigid_fz = 4.4180
inertie_mx = 88.464
zp_centre = -0.5181

with type_noe = 2:
rigid_fy = 5.8258
rigid_fz = 4.3912
inertie_mx = 88.651
zp_centre = -0.5247

Figure 3.20 - Ile de Ré Bridge, half-section of deck on a pier

It is noted that the second node processing mode provides results in general closer to those of reference, but that the first mode provides acceptable results. The irreducible differences are attributable to the approximation that is made by applying the theory of thin-walled profiles.

Examples

The example below that has 6 nodes, 2 dangling branches and 6 slabs is derived from that of Figure 3.7 (same coordinate reference frames, same contours, same stress calculation points).

Point 2 for tangential and normal stress calculation, however, is placed in the axis of the right web, rather than its exterior cladding.

Given their reasonable size, we chose to integrate the gussets at the nodes.

The presence of nodes 4 and 6 is indispensable, as without them, it would be impossible to determine the mean line in these "bends".

The mechanical characteristics and shear flow data provided by the example of Figure 3.7 are those that the GE5 module would calculate for this section, so modeled.

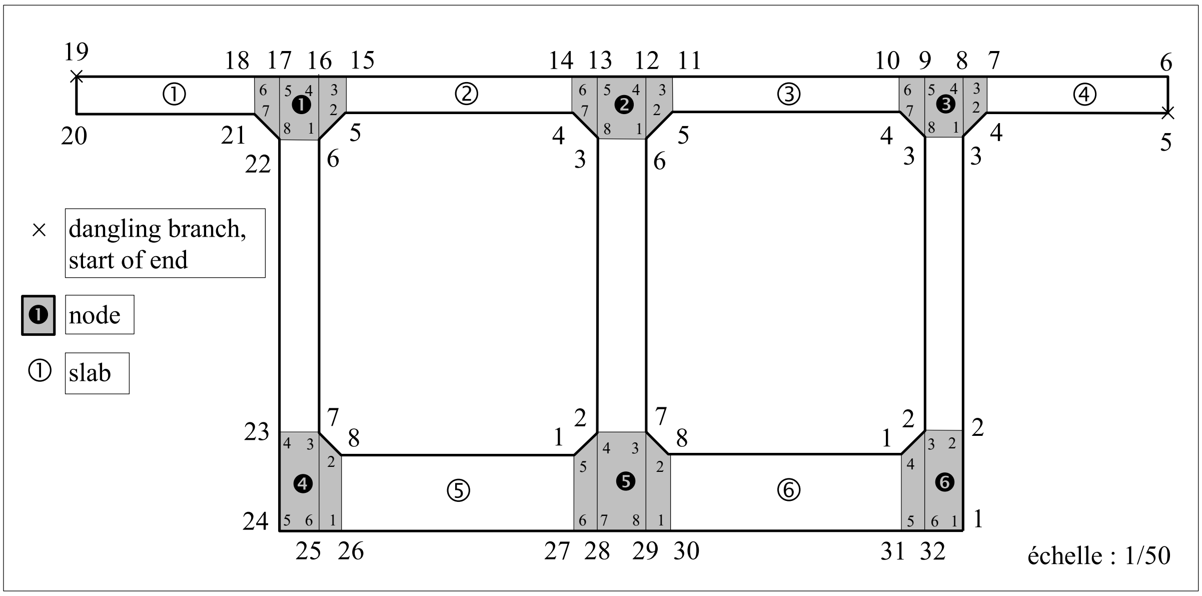


Figure 3.21 - Contoured section with morphology and slabs

```
$           n°_sigma n°_tau type_s orig_s type_sp type_gth nb_hou type_node
GENERALITES  4        4      1      0.0    1       1        6      2
...................................
SECTION CONTOURS ENTIERE  CAISSON1  2
$ numbers of contour points
32  8  8
$ numbers of top slab points
 6  8  8  6  8  8
$ exterior contour, the first point is the first point of node 6
$ points 5 and 19 are the first hanging branch extremities
 2.25 -3.00   1 6      2.25 -2.35   2 6      2.25 -0.40   1 3
 2.40 -0.25   2 3      3.60 -0.25   1 0      3.60  0.00   0 0
 2.40  0.00   3 3      2.25  0.00   4 3      2.00  0.00   5 3
 1.85  0.00   6 3      0.30  0.00   3 2      0.15  0.00   4 2
-0.15  0.00   5 2     -0.30  0.00   6 2     -1.85  0.00   3 1
-2.00  0.00   4 1     -2.25  0.00   5 1     -2.40  0.00   6 1
-3.60  0.00   1 0     -3.60 -0.25   0 0     -2.40 -0.25   7 1
-2.25 -0.40   8 1     -2.25 -2.35   4 4     -2.25 -3.00   5 4
-2.00 -3.00   6 4     -1.85 -3.00   1 4     -0.30 -3.00   6 5
-0.15 -3.00   7 5      0.15 -3.00   8 5      0.30 -3.00   1 5
 1.85 -3.00   5 6      2.00 -3.00   6 6
$ cutting 1
-0.30 -2.50   5 5     -0.15 -2.35   4 5     -0.15 -0.40   8 2
-0.30 -0.25   7 2     -1.85 -0.25   2 1     -2.00 -0.40   1 1
-2.00 -2.35   3 4     -1.85 -2.50   2 4
$ cutting 2
 1.85 -2.50   4 6      2.00 -2.35   3 6      2.00 -0.40   8 3
 1.85 -0.25   7 3      0.30 -0.25   2 2      0.15 -0.40   1 2
 0.15 -2.35   3 5      0.30 -2.50   2 5
$ top slab
$ the first point of top slab 1 is the 22nd point of the exterior contour
 22 0   17 0   18 0   19 0   20 0   21 0
$ the first point of top slab 2 is the 3rd point of the cutting 1
  3 1   13 0   14 0   15 0   16 0    6 1    5 1    4 1
  3 2    9 0   10 0   11 0   12 0    6 2    5 2    4 2
  5 0    6 0    7 0    8 0    3 0    4 0
 28 0    2 1    1 1    8 1    7 1   25 0   26 0   27 0
 32 0    2 2    1 2    8 2    7 2   29 0   30 0   31 0
$ calculation points of the normal stresses
-2.25 -3.00    2.25   -3.00   -3.60  0.00    3.60  0.00
$ calculation points of the tangential and normal stresses
 1.85 -3.00    2.00 1000.00    0.30  0.00    2.40  0.00
$ width of coating (calculated surface area/perimeter ratios)
7.20
```

Related commands

GENERALITES ; SECTION CONTOURS PARTIELLE

AFFECTATION SECTION ; EPAISSEURS PAROIS

3.7 - PARTIAL SECTION CONTOUR

SECTION CONTOURS PARTIELLE nom_section non_section_ref

$$\text{nb\_points\_r\_ext} \quad [\langle \text{nb\_points\_r\_evid} \rangle_{\text{nb\_evid}}]$$

$$[\langle \text{ordre\_contour} \quad \text{y\_point} \quad \text{z\_point} \rangle_{\text{nb\_points\_r\_ext}}]$$

$$[\langle [\langle \text{ordre\_contour} \quad \text{y\_point} \quad \text{z\_point} \rangle_{\text{nb\_points\_r\_evid}}] \rangle_{\text{nb\_evic}}]$$

$$\langle \text{y\_sigma} \quad \text{z\_sigma} \rangle_{\text{nb\_sigma}} \quad \langle \text{y\_tau} \quad \text{z\_tau} \rangle_{\text{nb\_tau}} \quad \text{larg\_revet}$$

Parameters

The nb_sigma, nb_tau and type_sp parameters are provided by the GENERALITES command.

- nom_section: section name, which cannot be assigned to another beam section;
- nom_section_ref: name of the CONTOURS ENTIERE section with nb-evid cavities, from which it is derived by modifying the coordinates of certain of its points.

The morphology of this section is done afresh, as is the definition of any slabs.

- nb_points_r_ext: number of redefined points on the outer contour, positive or zero;
- nb_points_r_evid: number of redefined points of a cavity, positive or zero.

The total number of redefined points must be positive.

- ordre_contour: sequence number of a contour point;
- y_point, z_point: new coordinates.

The order of introduction of redefined points on a contour is not imposed.

- y_sigma, z_sigma, y_tau, z_tau, larg_revet: see SECTION CONTOURS ENTIERE command.

Functions

This command provides a thin-walled section defined partially, by polygonal contours, with morphology.

Conditions of use

- ☐ The use of this command is optional.
- ☐ This command can only exist if at least one CONTOURS ENTIERE section has been defined.

Methodological advice

- ☐ Combined points being admitted, this technique can be used to change the appearance of a section (creation or deletion of gussets, for example).

Examples

Returning to the example in Figure 3.20, and changing the thickness of the lower slab from 0.50 m to 0.25 m.

The outer contour remains unchanged, but points 2 and 23 are redefined to preserve the orthogonality of cuts on nodes 4 and 6.

Point 2 for tangential and normal stress calculation will adapt to the new position of the center of gravity.

```
SECTION CONTOURS PARTIELLE  CAISSON2  CAISSON1
$ numbers of redefined points on contours
 2  4  4
$ redefined points on the exterior contour
 2  2.25 -2.60   23 -2.25 -2.60
$ redefined points on cutting 1
 1 -0.30 -2.75    2 -0.15 -2.60    7 -2.00 -2.60    8 -1.85 -2.75
$ redefined points on cutting 2
 1  1.85 -2.75    2  2.00 -2.60    7  0.15 -2.60    8  0.30 -2.75
$ calculation points of the normal stresses
-2.25 -3.00    2.25   -3.00   -3.60  0.00    3.60  0.00
$ calculation points of the tangential and normal stresses
 1.85 -3.00    2.00 1000.00    0.30  0.00    2.40  0.00
$ width of coating (calculated surface area/perimeter ratios)
7.20
```

Related controls

GENERALITES ; SECTION CONTOURS ENTIERE

AFFECTATION SECTION ; EPAISSEURS PAROIS

3.8 - WALL SECTION

SECTION PAROIS nom_section nb_points nb_parois type_epaisseurs

$$[\text{epaisseur\_commune}] \quad \langle \text{y\_point} \quad \text{z\_point} \rangle_{\text{nb\_points}}$$

$$\left\{ \begin{array}{l} \langle \text{point\_debu t} \quad \text{point\_fin} \quad \text{epaisseur\_debut} \quad \text{epaisseur\_fin} \rangle_{\text{nb\_parois}} \\ \langle \text{point\_debu t} \quad \text{point\_fin} \quad \text{epaisseur} \rangle_{\text{nb\_parois}} \\ \langle \text{point\_debu t} \quad \text{point\_fin} \rangle_{\text{nb\_parois}} \end{array} \right\}$$

$$\langle \text{y\_sigma} \quad \text{z\_sigma} \rangle_{\text{nb\_sigma}} \quad \langle \text{y\_tau} \quad \text{z\_tau} \rangle_{\text{nb\_tau}} \quad \text{larg\_revet}$$

Parameters

The nb_sigma, nb_tau and type_sp parameters are provided by the GENERALITES command.

- nom_section: section name, which cannot be assigned to another beam section;
- nb_points: number of junction points, at least 3;
- nb_parois: number of walls, at least 2;
- type_epaisseurs: type of variation of wall thicknesses, indicate:
  - 1 if at least one wall is of variable thickness (linearly),
  - 2 if all the walls are of constant thickness, but generally different,
  - 3 if all the walls have the same constant thickness;
- epaisseur_commune: wall thickness, if type_épaisseurs = 3;
- y_point, z_point: coordinates of a junction point;
- point_debut, point_fin: wall junction start and end point numbers, determining its direction of travel.

The plane frame formed must be connected; each junction point must be connected to at least one wall; two junction points cannot be connected by more than one wall; the lengths of the walls must be greater than 0.01.

- epaisseur_debut, epaisseur_fin, epaisseur: thickness at the beginning and the end of a wall if type_epaisseurs = 1, or thickness of a wall if type_epaisseurs = 2.

The wall thickness must be greater than 0.001.

- y_sigma, z_sigma, y_tau, z_tau: see SECTION CONTOURS ENTIERE command; the mean lines of the walls are the mean line of the section;
- larg_revet: coating width (possibly zero), to be deducted from the outer perimeter of the section before calculating the surface/outside perimeter ratio if type_sp is 1 (the outer perimeter is then equated to twice the sum of the lengths of the walls); value provided of the surface/outer perimeter of the section if type_sp is 2.

Functions

This command provides a thin-walled section defined directly by its mean line and the thickness of its walls.

Conditions of use

- The use of this command is optional.
- The application of the theory of thin-walled profiles assumes the section is transversely non-deformable and stable (n aligned walls forbidden).

Examples

The mean line of the box beam below is enclosed in a rectangle of 7.20 m x 2 m.

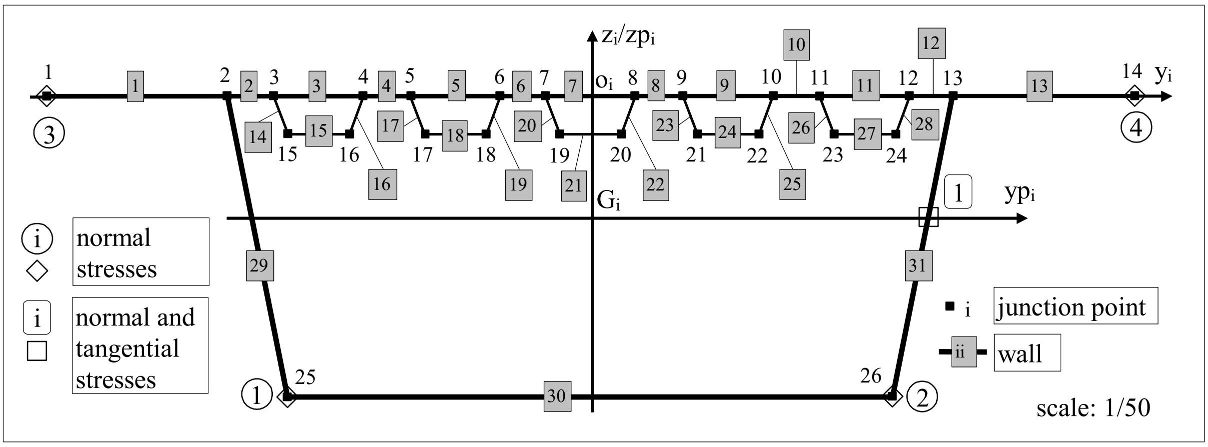


Figure 3.22 - Thin-walled section defined directly

```
$           n°_sigma n°_tau type_s orig_s type_sp type_gth nb_hou type_node
GENERALITES  4        1      1      0.0    1       1        0      2
...............................
SECTION PAROIS  CAISSON1  26  31  2
$ coordoninates of the junction points
-3.60  0.00   -2.40  0.00   -2.10  0.00   -1.50  0.00   -1.20  0.00
-0.60  0.00   -0.30  0.00    0.30  0.00    0.60  0.00    1.20  0.00
 1.50  0.00    2.10  0.00    2.40  0.00    3.60  0.00
-2.00 -0.25   -1.60 -0.25   -1.10 -0.25   -0.70 -0.25   -0.20 -0.25
 0.20 -0.25    0.70 -0.25    1.10 -0.25    1.60 -0.25    2.00 -0.25
-2.00 -2.00    2.00 -2.00
$ incidences of the walls and thicknesses
 1  2  0.015    2  3  0.015    3  4  0.015    4  5  0.015
 5  6  0.015    6  7  0.015    7  8  0.015    8  9  0.015
 9 10  0.015   10 11  0.015   11 12  0.015   12 13  0.015   13 14  0.015
 3 15  0.010   15 16  0.010   16  4  0.010    5 17  0.010   17 18  0.010
18  6  0.010    7 19  0.010   19 20  0.010   20  8  0.010    9 21  0.010
21 22  0.010   22 10  0.010   11 23  0.010   23 24  0.010   24 12  0.010
 2 25  0.020   25 26  0.020   26 13  0.020
$ calculation points of the normal stresses
 -2.00  -2.00     2.00 -2.00    -3.60  0.00     3.60  0.00
$ calculation points of the tangential and normal stresses
 2.00 1000.0
$ width of coating (calculated surface area/perimeter ratios)
7.20
```

Related commands

GENERALITES ; AFFECTATION SECTION ; EPAISSEURS PAROIS

3.9 - SECTION ASSIGNMENT

AFFECTATION SECTION nom_section nb_points

$$\langle \text{point\_fibr e} \rangle_{\text{nb\_points}}$$

Parameters

- nom_section: section name to be assigned;
- nb_points: number of reference fiber points to which it applies, at least 1;
- point_fibre: fiber reference point number; the order of introduction of points is not imposed.

Functions

This command generates a portion of the beam formwork, by assigning a generic section to a group of reference fiber points.

Conditions of use

- ❐ If there is only one section defined, do not assign it, it will be automatically assigned to all reference fiber points (default assignment).
- ❐ Otherwise, insert an AFFECTATION SECTION command per defined section, each point of the reference fiber will then receive a section assignment.

Examples

```
FIBRE REPERE  5  3
................
SECTION CONTOURS ENTIERE  CAISSON1  1
................
SECTION CONTOURS PARTIELLE  CAISSON2  CAISSON1
................
AFFECTATION SECTION  CAISSON1  2
1  5
AFFECTATION SECTION  CAISSON2  3
2  4  3
```

Related commands

FIBRE REPERE ; SECTION MASSIVE ; SECTION CONTOURS ENTIERE

SECTION CONTOURS PARTIELLE ; SECTION PAROIS ; TRONCONS

3.10 - SEGMENTS

TRONCONS nb_points

$\langle \text{point\_fibr e} \rangle_{\text{nb\_points}}$

Parameters

- nb_points: number of mean fiber "break" points, at least 1;
- point_fibre: number of the mean (or reference) fiber point where a break appears; a break point may not be introduced to the right of the start and end sections of the beam; introduction of numbers in ascending order is imposed; in the points list, a group of more than two consecutive numbers may not be introduced.

Functions

This command refers to the points of the mean beam fiber with path and/or slope discontinuities, to better control the calculation of its tangent vectors.

A single point is sufficient to indicate a break in slope. The tangent vector to the mean fiber is therefore the average of tangent vectors from the smoothing of segments located on the "left" and then the "right" of the angular point.

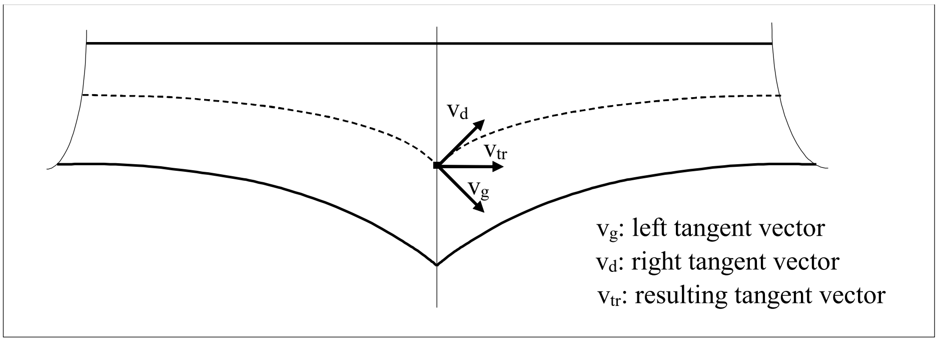


Figure 3.23 - Mean fiber, angular point

When the mean fiber path has a discontinuity due to the sudden change in the characteristics of the sections, we may designate two "close" consecutive breaking sections. Tangent vectors calculated at the left and right of this "transitional" item are then stored as parameters of the mean fiber.

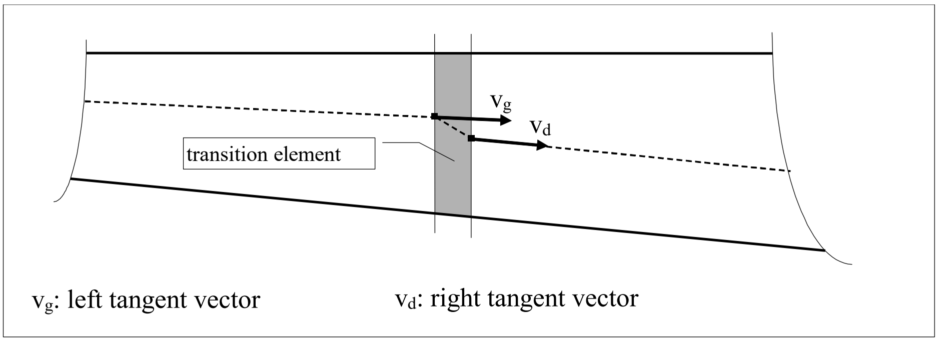


Figure 3.24 - Mean fiber, path discontinuity

Smoothing of the mean fibers of contoured section slabs follows the same rules as smoothing of the beam mean fiber; transition elements are assigned correction coefficients for zero Résal effect.

Conditions of use

- The use of this command is optional.

Methodological advice

- The length of the transition elements may be of the order of 1/10 of that of their adjacent elements.
- To create a transition element, just add a section to the left or right of the one with the "real" discontinuity.

Examples

The mean fiber below shows two path discontinuities and two angular points.

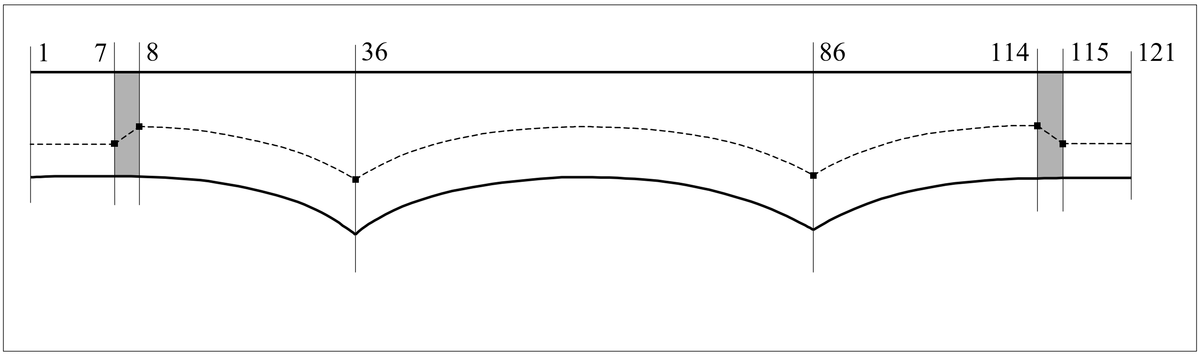


Figure 3.25 - Mean fiber, controlling discontinuities

```
FIBRE REPERE  121  3
..................
TRONCONS  6
  7    8 $ first transition element
 36   86 $ angular points
114  115 $ second transition element
```

Related commands

FIBRE REPERE ; AFFECTATION SECTION

3.11 - ARTICULATIONS

ARTICULATIONS nb_points

$$\langle \text{point\_fibre} \rangle_{\text{nb\_points}}$$

Parameters

- nb_points: number of internal articulations in the beam, at least 1;
- point_fibre: number of a point in the mean (or reference) fiber which contains an articulation; the order of introduction of numbers is not imposed.

Functions

This command provides the position of the internal articulations in the beam.

The corresponding sections will be simply identified by markers, on the formwork drawings.

Conditions of use

☐ The use of this command is optional.

Methodological advice

☐ The sections with articulations are not duplicated in the definition of the reference fiber, but are in the general mechanical model data.

Examples

```
ARTICULATIONS  2
24  96
```

Related controls

FIBRE REPERE

3.12 - WALL THICKNESS

EPAISSEURS PAROIS

$$\left\langle \left\langle \text{epaisseur\_ori} \quad \text{epaisseur\_ext} \right\rangle_{\text{nb\_tau}} \right\rangle_{\text{nb\_element}}$$

Parameters

The nb_tau parameter is supplied by the GENERALITES command; nb_elements is the number of beam elements.

- epaisseur_ori, epaisseur_ext: net thickness of a wall at the origin and at the end of an element, for a tangential and normal stress calculation point.

Functions

This command provides the net wall thicknesses to be taken into account for the calculation of tangential stresses; net thicknesses can differ from gross thicknesses if there are prestressing cables in ducts.

Duplication of values for the intermediate sections of the beam allows any sudden variations (cable stops) to be taken into account and for a calculation of stresses on both sides of each section to be performed.

Conditions of use

- ❐ Introduce only one command of this type.

Methodological advice

- ❐ See methodological advice for the SECTION MASSIVE command.

Examples

In the beam below, the first tangential and normal stress calculation point is in a slab of constant net thickness equal to 0.20 m, the second is located in a web whose net thickness goes from 0.50 m to 0.45 m, at element 3.

```
$           n°_sigma n°_tau type_s orig_s type_sp type_gth nb_hou type_node
GENERALITES  4         2      1      0.0    1        1        4      2
FIBRE REPERE  5  3 $ 5 sections, 4 elements
EPAISSEURS PAROIS
$ ---------- point 1 --------------------- point 2 -----------------
$ ori thickness  ext thickness  ori thickness   ext thickness
      0.20          0.20            0.50            0.50  $ element 1
      0.20          0.20            0.50            0.50  $ element 2
      0.20          0.20            0.50            0.45  $ element 3
      0.20          0.20            0.45            0.45  $ element 4
```

Related controls

GENERALITES ; FIBRE REPERE ; SECTION MASSIVE ; SECTION PAROIS

SECTION OUTLINES WHOLE; SECTION OUTLINES PARTIAL

3.13 - MATERIALS

MATERIAU nom_materiau nb_parametres

loi_fluage loi_module loi_retrait

module_reference coef_poisson poids_volumique coef_dilatation

$\langle \text{para\_supp} \rangle_{\text{nb\_parametres}}$

Parameters

- nom_materiau: name of the material, which cannot be assigned to another material of the model (of the beam, other beams and non-beam elements);
- nb_parametres: number of additional parameters associated with rheological behavior laws, indicate 3;
- loi_fluage: creep law number;
- loi_module: instantaneous Young's modulus variation law number;
- loi_retrait: shrinkage law number;
- module_reference: Young's modulus used in the absence of fine rheological calculation;
- coef_poisson: Poisson's ratio, must be greater than -1.0;
- poids_volumique: specific weight;
- coef_dilatation : thermal expansion coefficient;
- para_supp: additional parameter whose content depends on the rheological behavior laws chosen.

See Annex B for the meaning of the numbers of the rheological behavior laws and their additional parameters.

When an additional parameter is not used, enter a readable fictitious value (1.0).

The parameters module_reference, poids_volumique and coef_dilatation must be greater than 1.0E-8.

Functions

This command provides the basic beam materials, characterized by physical parameters and behavioral laws over time, for certain phenomena, set by the regulations.

These behavioral laws and their associated parameters are used by the PH3 module in CALCUL RHEOLOGIQUE FIN mode.

Conditions of use

- ☐ Provide at least one command of this type.
- ☐ The number of materials cannot be greater than the number of beam elements.

Methodological advice

- The beam material names can contain an identifier particle, that can be used to generate the names of other materials, if there are any copies of the beams in the general mechanical model data.
- We must find this particle in the standard cable and cable names (see CARACTERISTIQUES CABLES and TRACE CABLE commands).
- For steel, there is no secant Young's modulus; for concrete, enter a module_reference value corresponding to the secant modulus if it is intended to make calculations with this modulus; if not, choose the instantaneous modulus.
- When the PH3 module is working in CALCUL RHEOLOGIQUE FIN mode (see Chapter 8, DATE command), it always takes into account the instantaneous Young's modulus (para_supp$_1$); otherwise the module_reference is chosen, unless the EINSTANTANE option is chosen in the ACTIVER ELEMENTS command (see Chapter 8).

Examples

```
POUTRE  1     $ designated by particle P1 in the material names
...................
$ BPEL-compliant concrete
MATERIAU  P1BETON1  3
3                    $ creep law
14                   $ variation law of the instantaneous Young’s modulus
2                    $ retraction law
3.60E06              $ reference Young’s modulus
0.20                 $ Poisson coefficient
2.50                 $ unit weight
1.0E-05              $ thermal expansion coefficient
3.60E06              $ instantaneous modulus after 28 days
1.0                  $ unused parameter
0.95                 $ parameter KS of the regulation
```

Related controls

POUTRE ; FIBRE REPERE ; AFFECTATION MATERIAUX

CARACTERISTIQUES CABLES ; TRACE CABLE

3.14 - MATERIALS ASSIGNMENT

AFFECTATION MATERIAUX nb_troncons

$\langle \text{nom\_materiau} \quad \text{point\_fibre} \rangle_{\text{nb\_troncons-1}}$ nom_materiau

Parameters

- nb_troncons: number of beam segments formed of a single material, at least equal to the number of materials and at most equal to the number of beam elements;
- nom_materiau: name of a material;
- point_fibre: number of the end section of the segment to which it applies; the first segment begins at the first section and the last segment ends at the last section (not included); introduction of numbers in ascending order is imposed.

Functions

This command assigns the basic materials to the beam elements.

Conditions of use

- If there is a single material, do not assign it, it will automatically be assigned to all beam elements (default assignment).
- Otherwise, insert an AFFECTATION MATERIAUX command.
- Each material must be assigned to at least one element.

Examples

```
POUTRE  1     $ designated by particle P1 in the material names
...................
FIBRE REPERE  121  3
MATERIAU  P1BETON1  3
...................
MATERIAU  P1BETON2  3
...................
AFFECTATION MATERIAUX  3
P1BETON1  50  $ link 1, sections   1 to  50
P1BETON2  72  $ link 2, sections  50 to  72
P1BETON1      $ link 3, sections  72 to 121
```

Related commands

FIBRE REPERE ; MATERIAU

3.15 - CABLE CHARACTERISTICS

CARACTERISTIQUES CABLES nom_type
    sigma_initiale diametre_gaine section
    frot_courbes dev_parasite module_acier
    rentree_ancrage relax_1000h relax_3000h
    sigma_deformation sigma_rupture

Parameters

- type_name: name of the cable-type, which cannot be assigned to another cable-type of the model (of the beam and other beams);
- sigma_initiale: initial stress at an anchor;
- diametre_gaine: outside diameter of the duct;
- section: section;
- frot_courbes: coefficient of friction in curves, in degrees $^{-1}$;
- dev_parasite: coefficient of parasitic deviation;
- module_acier: Young's modulus of steel;
- rentree_ancrage: insertion of the anchor cone;
- relax_1000h, relax_3000h: relaxation at 1000 hours and 3000 hours, %;
- sigma_deformation :characteristic deformation stress ;
- sigma_rupture : guaranteed breaking stress.

For cables tensioned at both ends, sigma_initiale and rentree_ancrage are assumed identical at both anchors.

All these valuesmust be positive, except diametre_gaine, dev_parasite and recul_ancrage which may be zero; sigma_deformation must exceed sigma_initiale; sigma_rupture must exceed sigma_deformation.

Functions

This command provides a cable type, characterized by a set of physical parameters, which apply to certain prestressing cables.

Conditions of use

- ❐ Prestressing is optional, but if at least one cable-type is provided, at least one cable layout must be defined.
- ❐ Each cable type must be assigned to at least one cable layout, via the CABLE TRACE command.

Methodological advice

- Beam cable-type names can contain an identifier particle that can be used to generate other cable-type names if there are any copies of the beams in the general mechanical model data.
- We must find this particle in the names of materials and cables (see MATERIAU and TRACE CABLE commands).
- For external concrete prestressing, we can introduce both a zero duct diameter and a zero coefficient of parasitic deviation.
- When the friction losses before anchor setbacks of a cable tensioned at both ends are too low, it may be impossible to determine the zero displacement point; in this case, the cable must be tightened on one side only, or the coefficients of friction increased so as to obtain, for the cable stretched from one side, a total friction loss of at least 2/10000 of the initial tension.
- To transform a group of cables into passive steel bars, in order to perform non-linear calculations with the PH3 module, assign them a "very low" initial tension and a zero duct diameter.
- The coefficient of parasite deviation, denoted φ in the BPEL, is the product of μ and k in Eurocode 2.

Examples

```
POUTRE  1     $ designated by particle P1 in the typical cables names
..............
$ for definitive 27T15 cables, inside the concrete
$ ---------------------------------------------------
CARACTERISTIQUES CABLES  P1_27T15
144344.0      $ initial stress, 0.8 x guaranteed break stress
0.13          $ diameter of the sheath
4.05E-03      $ cross section
0.00314       $ friction coefficient in curves (degrees -1)
0.002         $ spurious deviation coefficient
1.9E07        $ Young's modulus of the steel
0.008         $ insertion of the anchoring cone
2.5           $ relaxation after 1,000 hours (%)
3.0           $ relaxation after 3,000 hours (%)
160380.0      $ characteristic deformation stress
180430.0      $ guaranteed break stress
```

Related commands

POUTRE ; MATERIAU ; TRACE CABLE

3.16 - CABLE LAYOUT

TRACE CABLE nom_cable nom_type m_tension nb_points categ m_calage

$$\left\{\begin{array}{l}\langle[\text{point\_fibre}]\quad \text{nb\_valeurs}\quad \text{s\_point}\quad \text{y\_point}\quad \text{z\_point}\quad [\text{dev\_ver}]\quad [\text{dev\_hor}]\rangle_{\text{nb\_points}} \\ \langle[\text{point\_fibre}]\quad \text{nb\_valeurs}\quad \text{s\_point}\quad \text{y\_point}\quad \text{z\_point}\quad [\text{rayon}]\rangle_{\text{nb\_points}}\end{array}\right\}$$

Parameters

The type_s and orig_s parameters are provided by the GENERALITES command; the curvilinear abscissa (approximating) are calculated, for the cables, along the reference fiber, in accordance with type_s.

- nom_cable: cable name, cannot be assigned to another cable of the model (of the beam and other beams);
- nom_type: cable-type name indicating its characteristics and appearing on a CARACTERISTIQUES CABLES command;
- m_tension: tensioning mode, indicate:
  - 1 if the cable is stretched at the start, 2 if the cable is stretched at the end;
  - 3 if the cable is stretched at both ends and released first at the start;
  - 4 if the cable is stretched at both ends and released first at the end;
- nb_points: number of definition points/poles, at least equal to 2 if categ = 1, and at least equal to 3 if categ = 2;
- categ: cable category, indicate:
  - 1 if the cable is defined by mandatory crossing points, with any imposed tangents;
  - 2 if the cable is defined by the nearby poles (forming a broken line of support), with any imposed circular connection radii;
- m_calage: longitudinal positioning mode of points/poles, indicate:
  - 1 if the curvilinear abscissa of each point/pole is measured relative to its own generic section;
  - 0 if the curvilinear abscissa of all cable points/poles are measured relative to an origin located at orig_s of section 1, in curvilinear abscissa;
  - a positive n value less than or equal to the number of points of the reference fiber, if the curvilinear abscissa of all cable points/poles are measured with respect to the section n;
- point_fibre: number of a curvilinear abscissa origin section, to be provided only if m_calage = 1; the order of introduction of the numbers is not imposed.
- nb_valeurs: number of actual values provided for a point/pole, indicate:
  - 3 in the case of a point without tangent, or a pole without connecting radius;
  - 4 in the case of a point with half tangent (a vertical or horizontal angle of deviation imposed), or a pole with connecting radius;
  - 5 in the case of a point with a tangent (two deviation angles imposed);
- s_point: curvilinear abscissa of a point/pole, measured relative to the point_fibre section if m_calage = 1, to the point at orig_s of section 1 if m_calage = 0, or to the m_calage section if m_calage is positive.

The generic reference frame $o_ix_iy_iz_i$ attached to a point/pole is the one whose curvilinear abscissa origin, measured relative to section 1: s_origine_1, is closest to the value of s_point, recalculated relative to the section 1: s_point_1.

The local abscissa of a point/pole in this reference frame is equated to s_point_1s_origine_1; it coincides with s_point if the section chosen for point/pole curvilinear abscissa origin is its closest section (in curvilinear abscissa).

If it is less than 0.02 m, it will be automatically reduced to 0.0 (resetting of points/poles too close to generic plane sections).

The curvilinear abscissa s_point_1 of cable points/poles must be strictly increasing, by at least 0.02 m.

No restrictions are imposed on the growth of curvilinear abscissa for different chosen origins if m_calage = -1 (cable setting on multiple origins).

- y_point, z_point: coordinates of a point/pole in its attached generic reference frame;
- dev_ver: "vertical" deviation angle, measured between the axis o_ix_i of the generic reference frame attached to a point and the projection of its tangent vector on the plane $x_io_iz_i$, counted positively from o_iz_i towards o_ix_i;
- dev_hor: "horizontal" deviation angle, measured between the axis o_ix_i of the generic reference frame attached to a point and the projection of its tangent vector on the plane $x_io_iy_i$, counted positively from o_ix_i towards o_iy_i.

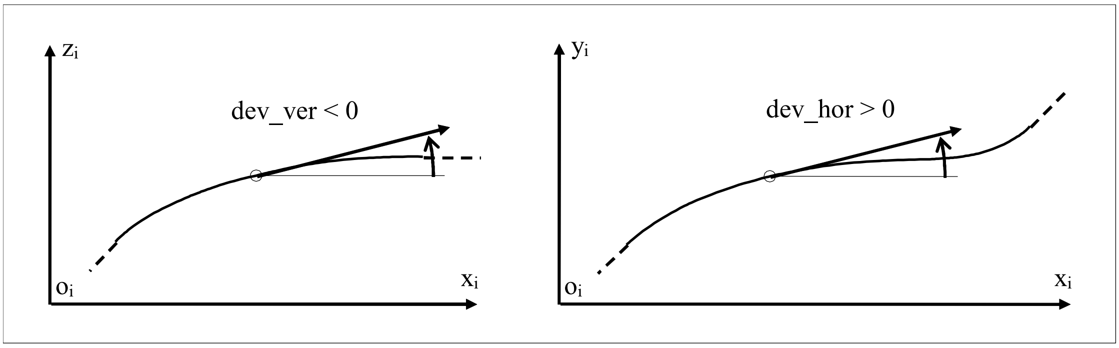


Figure 3.26 - Point cables, local tangent deviations

These optional angles are provided only for point cables (categ = 1), when nb_valeurs is greater than 3.

If nb_valeurs = 4, indicate a value with a 2 digit integer part, preceded by 1 if there is a vertical deviation, or 2 for a horizontal deviation, for example:

- 100.0 expresses a zero vertical deviation;
- 109.5 expresses a vertical deviation of 9.5 degrees;
- 215.4 expresses a horizontal deviation of 15.4 degrees.

If nb_valeurs = 5, indicate two effective deviation values (89.0 for the maximum absolute value) if the tangent is defined by deviations, otherwise indicate 90.0 90.0 if the tangent at the point i is carried by the segment [i1,i], or 90.0 90.0 if the tangent at the point i is carried by the segment [i,i +1].

- radius: the radius in space of the pole connecting circle, between 0.05 m and 1000.0 m.

This value is only provided for pole cables (categ = 2), when nb_valeurs = 4. It is not possible to impose a connecting radius to an anchor.

If this value is not provided (nb_valeurs = 3), a "minimum" connecting radius (0.09 m to 0.03 m depending on the case) is applied; it is a way to define a "nearly perfect" break.

If there is a conflict of connections, the values of radii will be adjusted upward or downward in order to keep the straight portions of significant length between an anchor and a circular arc, or between two arcs.

Functions

This command defines the geometry of a point, or a pole cable, its tensioning mode, and the name of the cable-type attached to it.

All pole cables will be transformed into point cables after determination of the points of tangency between straight and circular parts and replacement of circular parts by interpolation curves. All cable points will be finally replaced in their attached generic reference frames.

Conditions of use

- Prestressing is optional, but if at least one cable-type is provided, at least one cable layout must be defined.
- All cables must be described according to the direction of travel of the beam (Figure 3.4).
- Each cable, part of which may be outside the longitudinal domain of the beam, must pass through at least one generic section.

Methodological advice

- Beam cable names can contain an identifier particle that can be used to generate other cable names if there are copies of the beams in the general mechanical model data. We must find this particle in the names of materials and cable-types (see MATERIAU and CARACTERISTIQUES CABLES commands).
- Also include identifying particles of parts of the structure in the cable names (span, cantilever, web, etc.), allowing easy selection for completing or duplicating, if using the cable generator, or for them to be drawn; for example, all the cables from the left web contain the letter "G", to be replaced the letter "D" when creating cables for the right web by transverse symmetry.
- All generic section cable crossing points are not necessarily to be included in the definition.
- Any number of points can be added to the definition of a cable, between the generic sections it crosses.
- To impose a straight segment, two points can be provided with an aligned tangent direction (fictitious angular deviations of +90.0 and -90.0); this is the technique used to represent the straight sections of pole cables converted into point cables.

- Two consecutive points with imposed tangents suffice to define an "S" shaped curve, however, the angular deviation attributable to it is better controlled if we add the inflection point with its tangent (e.g.: "S" shaped curve obtained by two arcs of circles tangent at a point where the slope of the tangent is known).
- Avoid points too close together that can lead to significant and "hidden" angular deviations (kinks in the path).

Examples

The example below shows the first span of a variable height deck, built partly on falsework and partly by the cantilever method.

The construction prestressing is inside the concrete (point cables) and continuity prestressing is outside the concrete (pole cables); the pole cable concept adapts well to cables outside the concrete, but can be used for inside cables.

We define the half cable of the longest cantilever (which will be completed by symmetry) and a continuity cable covering the span.

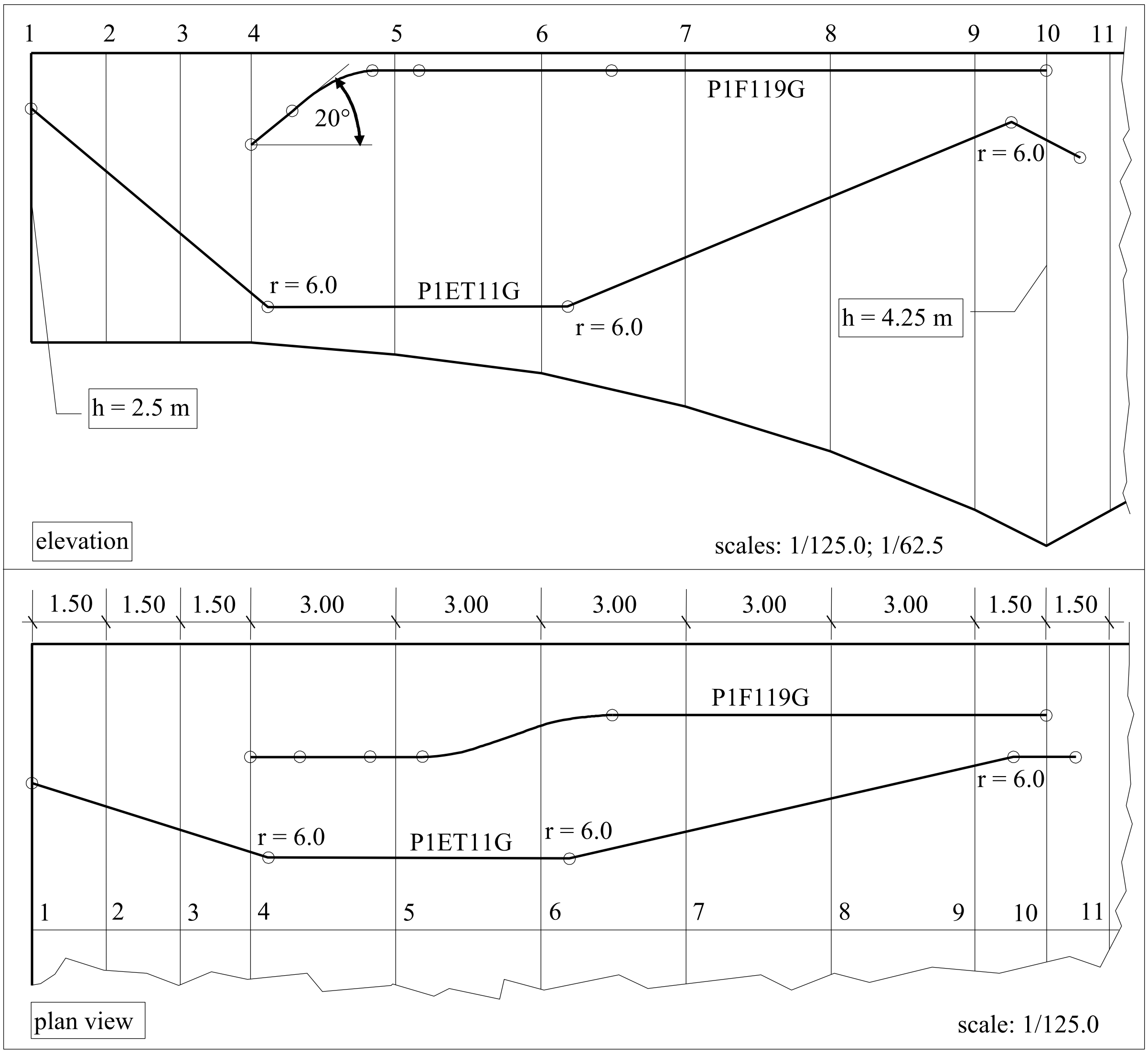


Figure 3.27 - Cable layout

```
POUTRE  1 $ designated by particule P1 in the typical cables/cables names
$           n°_sigma n°_tau type_s orig_s type_sp type_gth nb_hou type_node
GENERALITES  4        1      1      0.0     1       1        0      2
FIBRE REPERE  33  3      $ axis of the beam on the upper side
 0.00   5*0.0      1.50   5*0.0      3.00   5*0.0      4.50   5*0.0
 7.50   5*0.0     10.50   5*0.0     13.50   5*0.0     16.50   5*0.0
19.50   5*0.0     21.00   5*0.0     22.50   5*0.0     25.50   5*0.0
...................................................................
CARA CABL  P19T15S
..................
CARA CABL  P119T15S
..................
$ beam half-cable, longitudinally clamped on multiple sections
$           cable name type name  m_tension  n°_points  categ  clamping m
TRACE CABLE  P1F119G    P19T15S   3          6          1      -1
$ fiber point n°_values s_point y_point z_point dev_ver dev_hor
   4          5          0.00    3.00    -0.80   -20.00    0.00
   4          5          1.00    3.00    -0.44   -20.00    0.00
   4          5          2.50    3.00    -0.14     0.00    0.00
   5          5          0.50    3.00    -0.14     0.00    0.00
   6          5          1.50    3.75    -0.14    90.00   90.00
  10          5          0.00    3.75    -0.14   -90.00  -90.00
$ continuity cable, longitudinally clamped on multiple sections
$          cable name  type name m_tension  n°_points  categ  clamping m
TRACE CABLE  P1ET11G    P119T15S  3          5          2      -1
$ fiber point  n°_valuers  s_point  y_point  z_point  radius
   1           3           0.00     2.50     -0.50
   4           4           0.20     1.25     -2.20    6.00
   6           4           0.50     1.25     -2.20    6.00
   9           4           0.75     3.00     -0.60    6.00
  10           3           0.75     3.00     -0.90
```

Variant with half cable offset on section 10 (plane of symmetry), and continuity cable offset on section 1 which is the origin of the absolute curvilinear abscissa, the orig_s parameter of the GENERALITES command being zero.

```
TRACE CABLE  P1F119G  P19T15S   3  6  1  10
$ n°_values  s_point  y_point  z_point  dev_ver  dev_hor
  5          -16.50   3.00     -0.80    -20.00     0.00
  5          -15.50   3.00     -0.44    -20.00     0.00
  5          -14.00   3.00     -0.14      0.00     0.00
  5          -13.00   3.00     -0.14      0.00     0.00
  5           -9.00   3.75     -0.14     90.00    90.00
  5            0.00   3.75     -0.14    -90.00   -90.00
TRACE CABLE  P1ET11G  P119T15S  3  5  2   0
$ n°_values  s_point  y_point  z_point  radius
  3           0.00    2.50     -0.50
  4           4.70    1.25     -2.20    6.00
  4          11.00    1.25     -2.20    6.00
  4          20.25    3.00     -0.60    6.00
  3          21.75    3.00     -0.90
```

Related controls

POUTRE ; GENERALITES ; FIBRE REPERE ; MATERIAU

CARACTERISTIQUES CABLES

3.17 - END

FIN

Functions

This command marks the end of the beam data.

Conditions of use

- Must be the last line of the command file.

Examples

```
POUTRE  100
...........
...........
FIN
```

Related controls

POUTRE

Chapter 4

Cable generation

Introduction

This chapter describes the data required to complete the cabling of a (spatial) beam with redundant parts that have not been defined in the basic cables (missing parts of half defined symmetrical cables, cables that are obtainable by translating or symmetrizing entirely defined cables, etc).

The term "section" refers to a generic section, the term "reference frame" to a generic reference frame, and the term "local coordinates" to coordinates in multiple generic reference frames.

The MC1 module "handles" only cables with points (defined as such or resulting from the transformation, by the GE1 module, of cables with poles), with or without whole tangents (which can result from completion, by the GE1 module, of the half tangents of cables with points).

Data

Usually one file is produced per beam, its size is not limited.

Storage tables can accommodate up 2,500 fiber reference points, 20,000 cable names and 250,000 cable definition points (statically allocated memory); however, the maximum size of the problem to be processed is below these limits if cables are deleted or completed.

The input data order determines the order of execution of commands, the effect is cumulative.

With no end delimiter, each command in the MC1 module must begin on a new line and the first line break of the wording is to be observed. However, cutting data that follows a command header into lines, is unrestricted.

Analysis mode

The MC1 module performs an interpretive data analysis (coupled with a control) in a single pass, stopping at the first error.

Constant checking is performed to ensure that all "active"cables have correct and different names and that the geometry of completed cables, moved or copied by symmetry or translation, is consistent with the constraints imposed by the GE1 module (in particular, strictly increasing absolute curvilinear abscissa of points and crossing at least one section).

Processing sequence

If the MC1 module has not detected any error, it is verified that the beam has at least one cable of each predefined type (allocation of all the standard cables of the basic cabling to at least one cable path), and the modified cabling is recorded.

The MC2 module then runs, reproducing the content of the database, then the GE3 and GE5 modules run, as if the generated cables were introduced as data into the GE1 module.

Editing

The MC1 module only provides an echo of the analyzed commands.

The MC2 module provides a reminder in clear form of the basic beam data and its completed cabling, having undergone a first state of transformation.

The GE3 and GE5 modules provide the results of the various geometrical and mechanical processing.

The volume of these outputs is globally scalable.

Contents

4.1 - CABLES

CABLES no_poutre

Parameters

- no_poutre: the beam number of the beam whose cabling will be modified, from 1 to 9999, must correspond to a beam registered in the database by the GE1 module.

Functions

This command causes the rereading of a beam in the database and marks the beginning of the generation of the cabling data.

Conditions of use

- ☐ Must be the first line of the command file.

Examples

```
CABLES  100
...........
...........
FIN
```

Related controls

FIN

4.2 - DELETION

$$\text{EFFACEMENT}\quad\left\{\begin{array}{ll}\text{LISTE} & \text{nb\_cables}\\ \text{PARTICULE} & \text{psel}\end{array}\right\}$$

$$[\left\langle \text{nom\_cable}\right\rangle_{\text{nb\_cables}}]$$

Parameters

- nb_cables: number of cables to be deleted, if cables selected by list (LIST option), positive;
- psel: selection of a particle (similar to a name), if cables selected by particle (PARTICULE option); all cables whose names contain this particle at least once will be deleted;
- nom_cable: name of a cable to be deleted, if cables selected by list.

Functions

This command removes a group of cables.

Conditions of use

- The use of this command is optional.
- After running this command, the beam must have at least one cable of each of the predefined types (allocation of all the standard cables of the basic cabling to at least one cable path).

Examples

```
$ removal of two empty jackets
EFFACEMENT LISTE  2
P1F1GV1  P1F1GV2

$ removal of cables whose names contain the particle "GV"
EFFACEMENT PARTICULE  GV
```

4.3 - COMPLETION

$$\text{COMPLETION}\ \left\{\begin{array}{l}\text{ORIRELATIF}\\ \text{ORIABSOLU}\\ \text{EXTRELATIF}\\ \text{EXTABSOLU}\end{array}\right\}\ \left\{\begin{array}{ll}\text{LISTE} & \text{nb\_cables}\\ \text{PARTICULE} & \text{psel}\end{array}\right\}$$

$$[\langle \text{nom\_cable} \rangle_{\text{nb\_cables}}]$$

Parameters

- nb_cables: number of cables to be completed, if cables selected by list (LIST option), positive;
- psel: selection of a particle (similar to a name), if cables selected by particle (PARTICULE option); all cables whose names contain this particle at least once will be completed;
- nom_cable: name of a cable to be completed, if cables selected by list.

Functions

This command completes by longitudinal symmetry, a group of semi-defined cables, by their origin, if their second half has been defined (ORIRELATIF and ORIABSOLU options), or their end, if only their first half has been defined (EXTRELATIF and EXTABSOLU options).

If NB_POINTS is the initial number of points of a half cable, its completed counterpart will have $2 \times \text{nb\_points} - 1$ points.

The transformation can take place on curvilinear abscissa which are relative (ORIRELATIF and EXTRELATIF options), or absolute (ORIABSOLU and EXTABSOLU options).

In the first case, a search is done for each cable to be duplicated and for each point (which always has a reference frame and a local abscissa), for the section number associated with it "by symmetry" (which must exist), its local abscissa having changed sign. This type of transformation only doubles the effective curvilinear length of the cable if the distribution of the sections is "symmetrical" in their longitudinal effective zone.

In the second case, each cable to be completed is reset to absolute curvilinear abscissa, with respect to Section 1, completed by symmetry, then reset on the sections closest to its points. This type of transformation doubles the effective curvilinear length of the cables regardless of the distribution of the sections in their longitudinal zone of influence.

In both cases, the local coordinates y and z are retained and the vertical and horizontal deviations of any tangents (real or fictional) change sign.

Conditions of use

- The use of this command is optional.

Methodological advice

- The choice of the transformation mode (relative or absolute) is immaterial if the distribution of sections, in curvilinear abscissa, is symmetrical (within 0.02 m) in the longitudinal effective zone of the completed cables.
- The cable tensioning modes remain unchanged in the transformation. Introduce, for the half-cables, the tensioning modes of the completed cables, or if needed restore them using the TENSION command.

Examples

Cables and C1R and C1A below, schematically represented viewed from above, are completed by symmetry with respect to their ends, with only their first halves being defined. Cable C1R is completed in relative mode, and cable C1A in absolute mode; the results are not the same because the longitudinal distribution of the sections is not symmetrical in the longitudinal effective zone of the cables (sections 10 to 19).

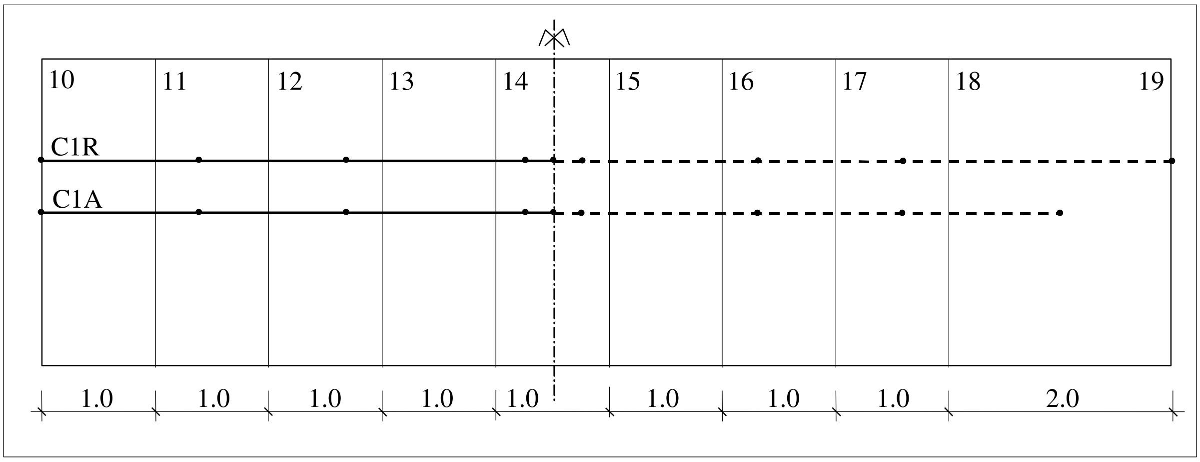


Figure 4.1 - modes for completing by symmetry

```
$ 5-point cable defined by its first half
TRACE CABLE  C1R  C12T13S  3  5  1  -1
10  5   0.00   2.50  -1.25  -20.0    0.0
11  5   0.40   2.50  -0.25    0.0    0.0
13  5  -0.30   2.50  -0.25    0.0    0.0
14  5   0.25   2.50  -0.25    0.0    0.0
14  5   0.50   2.50  -0.25    0.0    0.0

$ 9-point cable resulting from the execution of the command
$ COMPLETEMENT EXTRELATIF PARTICULE  C1R
TRACE CABLE  C1R  C12T13S  3  9  1  -1
10  5   0.00   2.50  -1.25  -20.0    0.0
11  5   0.40   2.50  -0.25    0.0    0.0
13  5  -0.30   2.50  -0.25    0.0    0.0
14  5   0.25   2.50  -0.25    0.0    0.0
14  5   0.50   2.50  -0.25    0.0    0.0 $ symmetry drawing
15  5  -0.25   2.50  -0.25    0.0    0.0
16  5   0.30   2.50  -0.25    0.0    0.0
18  5  -0.40   2.50  -0.25    0.0    0.0
19  5   0.00   2.50  -1.25   20.0    0.0
```

```
$ 5-point cable defined by its first half
TRACE CABLE  C1A  C12T13S  3  5  1  -1
10  5   0.00   2.00  -1.25  -20.0    0.0
11  5   0.40   2.00  -0.25    0.0    0.0
13  5  -0.30   2.00  -0.25    0.0    0.0
14  5   0.25   2.00  -0.25    0.0    0.0
14  5   0.50   2.00  -0.25    0.0    0.0

$ 9-point cable resulting from the execution of the command
$ COMPLETEMENT EXTABSOLU PARTICULE  C1A
TRACE CABLE  C1A  C12T13S  3  9  1  -1
10  5   0.00   2.00  -1.25  -20.0    0.0
11  5   0.40   2.00  -0.25    0.0    0.0
13  5  -0.30   2.00  -0.25    0.0    0.0
14  5   0.25   2.00  -0.25    0.0    0.0
14  5   0.50   2.00  -0.25    0.0    0.0 $ symmetry drawing
15  5  -0.25   2.00  -0.25    0.0    0.0
16  5   0.30   2.00  -0.25    0.0    0.0
18  5  -0.40   2.00  -0.25    0.0    0.0
18  5   1.00   2.00  -1.25   20.0    0.0
```

Related commands

TENSION

4.4 - TRANSLATION

$$\text{TRANSLATION}\left\{\begin{matrix}\text{LONRELATIVE} & \delta n\\ \text{LONABSOLUE} & \delta s\\ \text{TRANSVERSALE} & \delta y\\ \text{VERTICALE} & \delta z\end{matrix}\right\}\left\{\begin{matrix}\text{COPIE}\\ \text{MOUVEMENT}\end{matrix}\right\}\left\{\begin{matrix}\text{LISTE} & \text{nb\_cables} & \\ \text{PARTICULE} & \text{psel} & [\text{psub}]\end{matrix}\right\}$$

$$[\langle \text{nom\_cable\_s} \quad [\text{nom\_cable\_c}]\rangle_{\text{nb\_cables}}]$$

Parameters

- δn: longitudinal offset measured by number of sections, in the case of relative longitudinal translation (LONRELATIVE option); integer nonzero algebraic value to be added to all the reference frame numbers of the points of the cables to be translated; this value should not cause cable "exits" from the longitudinal region of the beam (non existing reference frames);
- δs: longitudinal offset measured in curvilinear abscissa, in the case of absolute longitudinal translation (LONABSOLUE optional); integer nonzero algebraic value to be added to the absolute abscissa (recalculated with respect to section 1) of the points of the cables to be translated; this value can cause "partial exits" from the longitudinal region of the beam; the translated cables are readjusted on the closest sections of their points of definition;
- δy, δz: transverse or vertical offset, in the case of transverse (TRANSVERSE option) or vertical translation (VERTICALE option); real nonzero algebraic value to be added respectively to the local coordinates y or z of the points of the cables to be translated;
- nb_cables: number of cables to be translated, if cables selected by list (LIST option), positive;
- psel: selection of a particle (similar to a name), if cables selected by particle (PARTICULE option); all cables whose names contain this particle at least once will be translated;
- psub: substitution of a particle (similar to a name), if cables selected per particle, and copy transformation (COPIE option); cable names generated are obtained by replacing, in those of the selected cables, the first occurrence of the psel particle, by psub;
- nom_cable_s: name of a cable to be translated, in the case of selection of cables by list;
- nom_cable_c: name of cable to be created, by translation of the cable nom_cable_s, in the case of a copy transformation, and selection of cables by list.

Functions

This command allows translation of a group of cables, longitudinally (LONRELATIVE or LONABSOLUE options), transversely (TRANSVERSALE option) or vertically (VERTICALE option).

Longitudinal translation can take place in relative curvilinear abscissa (LONRELATIVE option, parameter δn) or absolute abscissa (LONABSOLUE option, parameter δs).

In the first case, the effective curvilinear cable length is maintained only if the distribution of the sections is "identical" in their longitudinal effective zones (initial and received).

In the second case, the effective curvilinear cable length is maintained, regardless of the distribution of the sections in their longitudinal effective zones.

The cable point numbers and the non relevant local coordinates (y and z in the case of longitudinal translation, x and z in case of transverse translation, x and y in case of vertical translation) are retained, as well as any tangent deviations (real or fictional).

If the transformation is of copy type, the selected cables are duplicated; if the transformation is of movement type (MOUVEMENT option), the selected cables are moved.

Conditions of use

- The use of this command is optional.

Methodological advice

- The choice of the longitudinal transformation mode (relative or absolute) is immaterial if the distribution of sections, in curvilinear abscissa, is identical (within 0.02 m) in the longitudinal effective zone of the cables to be translated, and the cables produced by translation, and if the value of the parameter δs is an integer number of section intervals.
- The cable tensioning modes remain unchanged in the transformation, if needed, change the cable tensioning modes resulting from the transformation using the TENSION command.

Examples

The T1O cable below, schematically represented viewed from above, is copied longitudinally by translation, in relative mode (result: T1R cable) and in absolute mode (result: T1A cable, transversely offset for clarity of the figure); the amplitude of the translation in both cases corresponds to an integer number of intervals of equidistant sections, but with the length of the beam section located between sections 18 and 19 being 2.0 m, the T1R and T1A and cables have different lengths.

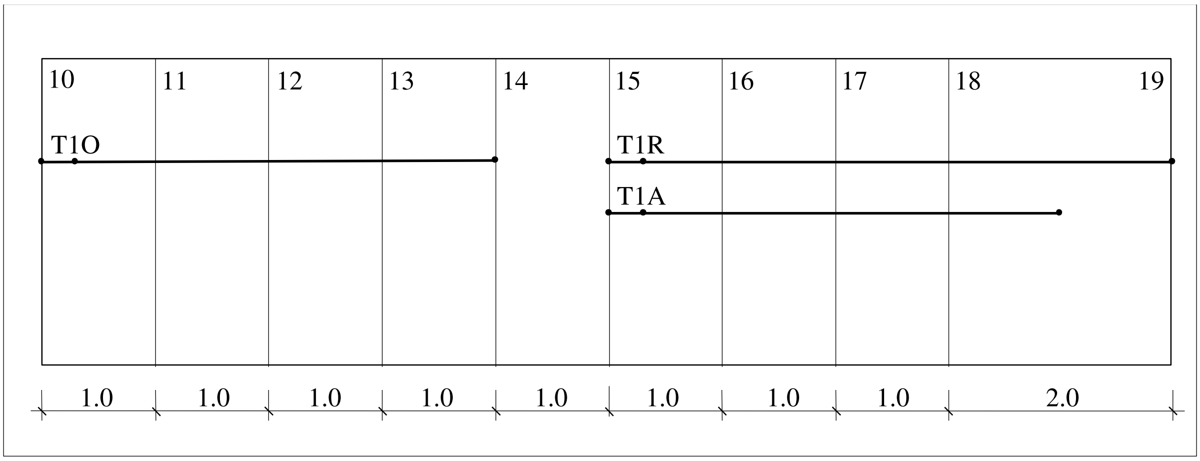


Figure 4.2 - copy modes by longitudinal translation

```
$ 3-point cable defined in its entirety
TRACE CABLE  T1O  C12T13S  3  3  1  -1
10  5   0.00   2.50  -1.25  -20.0    0.0
10  5   0.30   2.50  -0.75  -20.0    0.0
14  5   0.00   2.50  -0.25    0.0    0.0

$ 3-point cable resulting from the execution of the command
$ TRANSLATION LONRELATIVE  5  COPIE  LISTE  1
$ T1O T1R
TRACE CABLE  T1R  C12T13S  3  3  1  -1
15  5   0.00   2.50  -1.25  -20.0    0.0
15  5   0.30   2.50  -0.75  -20.0    0.0
19  5   0.00   2.50  -0.25    0.0    0.0

$ 3-point cable resulting from the execution of the command
$ TRANSLATION LONABSOLUE  5.0  COPIE  PARTICULE  T1O  T1A
$ TRANSLATION TRANSVERSALE  -0.50  MOUVEMENT  LISTE  1
$ T1A
$ TRANSLATION VERTICALE  0.10  MOUVEMENT  PARTICULE  T1A
TRACE CABLE  T1A  C12T13S  3  3  1 -1
15  5   0.00   2.00  -1.15  -20.0    0.0
15  5   0.30   2.00  -0.65  -20.0    0.0
18  5   1.00   2.00  -0.15    0.0    0.0
```

Related commands

TENSION

4.5 - SYMMETRY

$$\text{SYMETRIE}\left\{\begin{matrix}\text{LONRELATIVE} & \text{npv}\\ \text{LONABSOLUE} & \text{spv}\\ \text{TRANSVERSALE} & \text{ypv}\end{matrix}\right\}\left\{\begin{matrix}\text{COPIE}\\ \text{MOUVEMENT}\end{matrix}\right\}\left\{\begin{matrix}\text{LISTE} & \text{nb\_cables} & \\ \text{PARTICULE} & \text{psel} & [\text{psub}]\end{matrix}\right\}$$

$$[\langle \text{nom\_cable\_s} \quad [\text{nom\_cable\_c}]\rangle_{\text{nb\_cables}}]$$

Parameters

The parameter orig_s is supplied by the GENERALITES command (see Chapter 3).

- npv: section number serving as a pivot for the transformation; in the case of relative longitudinal symmetry (LONRELATIVE option), indicate:
  - a positive value actually designating a section;
  - or a negative value, i, denoting the plane midway between sections i and i+1;
- spv: curvilinear abscissa of the transformation pivot, measured relative to an origin located at orig_s of section 1, in the case of absolute longitudinal symmetry (LONABSOLUE optional); this does not necessarily coincide with a point of the reference fiber;
- ypv: offset of local z symmetry axes (possibly zero), with respect to generic z axes, in the case of transverse symmetry (TRANSVERSE option);
- nb_cables: number of cables to be symmetrized, if cables selected by list (LIST option), positive;
- psel: selection of particle (similar to a name), if cables selected by particle (PARTICULE option); all cables whose names contain this particle at least once will be symmetrized;
- psub: substitution of a particle (similar to a name), if cables selected by particle, and copy transformation (COPIE option); cable names generated are obtained by replacing, in those of the selected cables, the first occurrence of the psel particle, by psub;
- nom_cable_s: name of a cable to be symmetrized, if cables selected by list;
- nom_cable_c: name of the cable to be created, by symmetrization of the cable nom_cable_s, in the case of a copy transformation, and cable selection by list.

Functions

This command is to symmetrize a group of cables, longitudinally (LONRELATIVE or LONABSOLUE options), or transversely (TRANSVERSALE option).

Cable point numbers are retained.

The longitudinal symmetry may occur in relation to a relative curvilinear abscissa (LONRELATIVE option npv setting) or absolute abscissa (LONABSOLUE option spv parameter) fixed pivot.

In the first case, a search is done for each cable and for each point to be symmetrized (which always has a reference frame and a local abscissa), for the section number associated with it "by symmetry" (which must exist), its local abscissa having changed sign. This type of transformation only preserves the effective curvilinear length of the cable, if the distribution of the sections is "symmetrical" in their longitudinal effective zones (initial and received).

In the second case, each cable to be symmetrized is reset to absolute curvilinear abscissa, with respect to Section 1, symmetrized, then reset on the sections closest to its points. This type of transformation preserves the effective curvilinear length of the cables, regardless of the distribution of the sections in their longitudinal effective zones.

In both cases, the local coordinates y and z are retained and the vertical and horizontal deviations of any tangents (real or fictional) change sign.

In the case of transverse symmetry, reference frame numbers, local abscissa, the local z-coordinates and vertical deviations of any tangents (real or fictional) are stored; local coordinates are symmetrized, and signs of horizontal tangent deviations are changed.

If the transformation is of copy type, selected cables are duplicated; if the transformation is of movement type (MOUVEMENT option), the selected cables are moved.

Conditions of use

- The use of this command is optional.

Methodological advice

- The choice of the transformation mode (relative or absolute) is immaterial if the distribution of sections, in curvilinear abscissa, is symmetrical (within 0.02 m) in the longitudinal effective zone of the completed cables, and cables produced by symmetry, and if the spv parameter of the transformation fixes the pivot on a section or midway between two sections.
- The cable tensioning modes remain unchanged in the transformation, if needed, change the cable tensioning modes resulting from the transformation using the TENSION command.

Examples

The S1O cable below, schematically represented viewed from above, is copied longitudinally by symmetry, in relative mode (result: S1R cable) and in absolute mode (result: S1A cable, transversely offset for clarity of the figure); the symmetry pivot is fixed in both cases midway between sections 14 and 15, but with the length of the beam section located between sections 18 and 19 being 2.0 m, the S1R and S1A cables have different lengths.

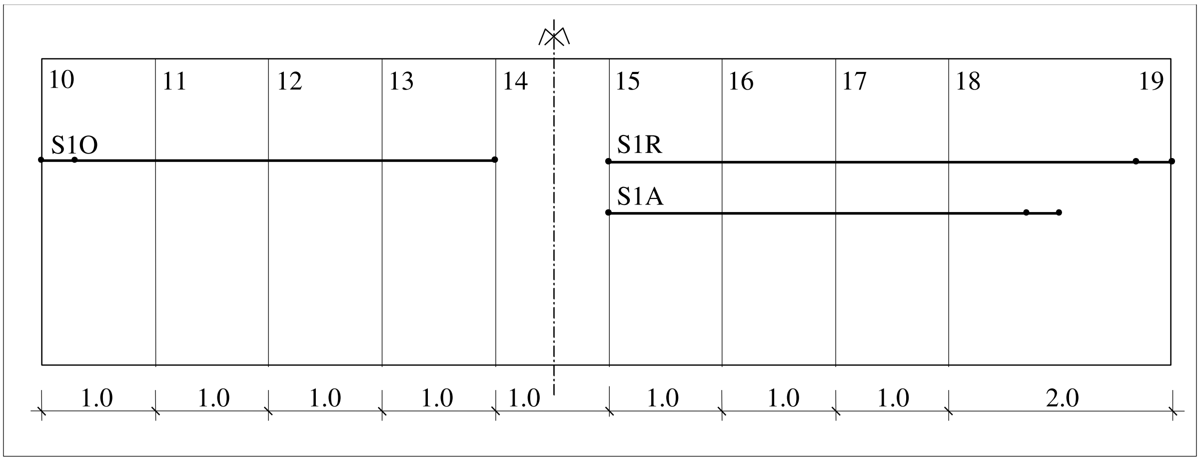


Figure 4.3 - copy modes for longitudinal symmetry

```
$ 3-point cable defined in its entirety
TRACE CABLE  S1O  C12T13S  3  3  1  -1
10  5   0.00   2.50  -1.25  -20.0    0.0
10  5   0.30   2.50  -0.75  -20.0    0.0
14  5   0.00   2.50  -0.25    0.0    0.0

$ 3-point cable resulting from the execution of the command
$ (the code -15 designates the "section" halfway between 14 and 15)
$ SYMETRIE LONRELATIVE  -15  COPIE  LISTE  1
$ S1O  S1R
TRACE CABLE  S1R  C12T13S  3  3  1  -1
15  5   0.00   2.50  -0.25    0.0    0.0
19  5  -0.30   2.50  -0.75   20.0    0.0
19  5   0.00   2.50  -1.25   20.0    0.0

$ 3-point cable resulting from the execution of the commands
$ (where sections 1 to 10 are assumed to be regularly separated by 1.0 m
$ and the parameter orig_s of the GENERALITES command is assumed to be nil,
$ the pivot of the transformation is 13.5 m from the start of the beam)
$ SYMETRIE LONABSOLUE  13.5  COPIE  PARTICULE  S1O  S1A
$ TRANSLATION TRANSVERSALE  -0.50  MOUVEMENT  PARTICULE  S1A
TRACE CABLE  S1A  C12T13S  3  3  1  -1
15  5   0.00   2.00  -0.25    0.0    0.0
18  5   0.70   2.00  -0.75   20.0    0.0
18  5   1.00   2.00  -1.25   20.0    0.0
```

Related commands

TENSION

4.6 - TENSION

$$\text{TENSION} \quad \begin{Bmatrix} \text{LISTE} & \text{nb\_cables} \\ \text{PARTICULE} & \text{psel} \end{Bmatrix} \quad \text{m\_tension}$$

$$[\langle \text{nom\_cable} \rangle_{\text{nb\_cables}}]$$

Parameters

- nb_cables: number of cables to be processed, if cables selected by list (LIST option), positive;
- psel: selection of a particle (similar to a name), if cables selected by particle (PARTICULE option); all cables whose names contain this particle at least once will be processed;
- m_tension: new tensioning mode, to be assigned to all selected cables (see Chapter 3, TRACE CABLE command, parameter m_tension);
- nom_cable: name of a cable to be processed, in the case of cables selected by list.

Functions

This command assigns a common value of tensioning mode to a group of cables.

Conditions of use

- The use of this command is optional.

Examples

```
$ change of tensioning mode of two cables
$ produced by a copy by longitudinal symmetry
TENSION LISTE  2  4
S1R  S1A

$ equivalent commands
TENSION PARTICULE  S1R  4
TENSION PARTICULE  S1A  4
```

Related commands

COMPLETEMENT ; TRANSLATION ; SYMETRIE

4.7 - END

FIN

Functions

This command marks the end of beam cabling data generation.

Conditions of use

- It should be the last line of the command file.

Examples

```
CABLES  100
...........
...........
FIN
```

Related commands

CABLES

Chapter 5

Drawing

Introduction

This chapter describes the additional data to be produced for a two-dimensional graphical representation of a spatial beam (selecting sections, cables and formwork segments to be drawn, positioning projection planes, execution of drawings).

"Raw" images can be obtained (GE2 module) using only the recorded data slightly reworked by the GE1 module, or "fine" images (GE4 module) representing the data and certain results produced by the GE3 module during their full geometric treatment (mean lines of contoured sections with morphology, mean fiber of the beam, smoothed cable routes). Raw representation is only accessible if the beam has been successfully registered by the GE1 module and if the GE3 module has detected at least one error.

To get a good graphical representation of a curved beam, there are two solutions:

- cut the beam at some of its sections and project the segments obtained on carefully selected planes, following its curvature;
- "stretch" the reference fiber along the O_bX_b axis of the reference frame of definition, effect a "temporary" recording by the GE1 module, and then project the beam as "developed" on two orthogonal planes to draw it, in whole or in segments.

The $X_bO_bY_b$ *plane of the beam reference frame, called the "horizontal plane", is associated with the notion of "plan view", while the* $X_bO_bZ_b$ *plane, called the "vertical plane", is associated with the notion of "elevation view". By extension, the generic reference frame* o_iy_i *axes are considered to be "local horizontals" and the* o_iz_i *axes "local verticals" (Figure 3.4).*

Scales

The GE2 and GE4 modules work in the maximum scales imposed, and may be different in the two directions. When the display area size of the chosen equipment (screen or plotter) proves insufficient to accommodate some of the drawings, the corresponding scale factors will be adjusted upward and rounded, otherwise they will be kept.

Projection planes

The projection planes have a plan or elevation view attribute, determining the formwork shell type (horizontal or vertical) to appear on the projected drawings; they may be linked to the beam, based on two points of its fiber reference frame and certain beam or generic reference frame axes, or positioned directly, in the beam reference frame.

Cable families to be drawn

Cables can be selected in full, by particle, or by name list.

Drawings

The GE2 and GE4 modules produce drawings of sections, projected formwork, and projected cabling.

The GE2 module represents the sections with their generic reference frame, the formwork with the reference fiber, and the cables by broken lines based on their points of definition, with arrows representing the tangent vectors. The GE4 module represents sections using their main inertia reference frame, the formwork with the beam mean fiber, and the cables using smoothed curves tabulated at steps of 0.25 m.

Data

A file must be created for each series of drawings to be produced, its size is not limited.

Data can be entered in any order (except for commands at the start and end of the file), forward references are possible (drawing a family of yet undefined cables, drawing cables projected on a yet undefined plane, etc.). The input order of drawing commands, however, determines their execution order and the order of production of drawings by category (sections, formwork, cabling).

With no end delimiter, each command in the GE2 or GE4 modules must begin on a new line and the first line break of the wording is to be observed. However, cutting data that follows a command header into lines, is unrestricted.

Analysis mode

The GE2 and GE4 modules perform compilation type data analysis (coupled with a control), by hierarchical levels and successive scans of the entire file, from the broadest to the finest, stopping at the level where the first error occurs.

The number of possible errors that can be detected is not limited for a level, but the detection of an error of a certain level of severity, for a command, may prevent the search for other errors on the same level.

Treatment sequence

If the GE2 or GE4 module has detected no errors, drawings are produced sequentially in this order: drawings of sections, formwork, cabling.

If the chosen equipment is interactive (screen), breakpoints are provided for observing the drawings (zoom and cropping), otherwise (plotter or similar), drawings accumulate in a file.

Editing

The GE2 and GE4 modules provide a reminder of the data entered and some elements from their interpretation; the volume of these outputs is not scalable.

Contents

5.1 - DESBRUT / DESFIN

$$\left\{\begin{matrix}\text{DESBRUT}\\ \text{DESFIN}\end{matrix}\right\} \quad \text{no\_poutre}$$

Parameters

- no_poutre: number of the beam to be drawn, from 1 to 9999, must correspond to a beam registered in the database by the GE1 module.

Functions

This command causes the rereading of a beam in the database and marks the beginning of a drawing dataset for it.

A representation can be obtained from the raw data from the GE1 module, possibly modified by the MC1 module (DESBRUT option, GE2 module), or a finer drawing can be obtained including the data and certain results of the GE3 module (DESFIN option, GE4 module).

Conditions of use

- Must be the first line of the command file.

Methodological advice

- GE4 is the most used module, especially for cables, for which it provides a representation close to the actual cable route.
- Use of the GE2 module is only authorized when the GE3 module has detected at least one error (determination of the mean line of certain contoured sections with a morphology that is not possible, smoothing the mean fiber of the beam or certain cables that are not possible, etc.).

Examples

```
DESBRUT  100
............
............
FIN
```

```
DESFIN  100
...........
...........
FIN
```

Related commands

DESSIN SECTIONS ; DESSIN FIBRE ; DESSIN FAMILLE CABLES ; FIN

5.2 - STANDARD SCALES

ECHELLES STANDARD echl echh

Parameters

- echl: imposed width scale factor (generic o_iy_i axes for drawings of segments, mean directions of development of the beam elements for formwork or cabling drawings);
- echh: imposed height scale factor (generic o_iz_i axes for drawings of sections, different orthogonal directions reflecting plan or elevation view, according to the type of projection planes selected, for formwork or cabling drawings).

These values must be greater than or equal to 0.1; enter 50 or 50.0 to express a scale of 1:50;

When either of these scale factors causes an overflow of the display area of the chosen equipment (screen or plotter), the one that causes the most severe overflow is adjusted to the higher value that allows to best fill the available space (in that direction); the other scale factor is then adjusted to maintain the ratio of the initial values; the two scale factors thus recalculated are finally reset to the nearest higher value multiple of 5.0.

For example, if the initial values are: echl = 100.0 and echh = 50.0, if there is overflow in both directions and if the minimum theoretical values required to fill the display area are: echl = 133.0 and echh = 77.0, the height overflow is the most severe, since 77.0/50.0 > 133.0/100.0; the corresponding factor (77.0) is retained and echl is changed from 133.0 to 77.0×2.0 = 154.0; the final adjustment of these values gives: echl = 155.0 and echh = 80.0; the initial ratio of scale factors (2.0) is therefore retained.

Functions

This command sets the scale factors to be taken into account for all drawing commands without specifying scale.

Conditions of use

- The use of this command is mandatory if introducing at least one drawing command without a specified scale, optional otherwise.

Methodological advice

- If the display area must be systematically filled in at least one direction, enter echl and echh values that are indicative but low; for example, code echl = 2.0 and echh = 1.0 if you wish to amplify the height dimensions of the drawings by a factor of 2.0 relative to their width dimensions.
- Take care not to thoughtlessly fill the display area of roller plotters, which can be large (several meters "width" for A0^+ format).

Examples

```
$ for screens or A4 or A3 printers, indicative scales
ECHELLES STANDARD  2.0  1.0

$ for roller plotters, imposed scales must be respected
$ as far as possible
ECHELLES STANDARD  100.0  50.0
```

Related commands

DESSIN SECTIONS ; DESSIN FIBRE ; DESSIN FAMILLE CABLES

5.3 - PROJECTION

$$\text{PROJECTION}\quad \left\{\begin{array}{l}\text{PLAN}\\ \text{ELEVATION}\end{array}\right\}\quad \left\{\begin{array}{l}\left\{\begin{array}{l}\text{GLOBAL}\\ \text{LOCAL}\end{array}\right\}\ \text{plan}\quad \text{sdeb}\quad \text{sfin}\\ \text{QUELCONQUE}\quad \text{plan}\quad \text{vnx}\quad \text{vny}\quad \text{vnz}\end{array}\right\}$$

Parameters

- plane: name of the projection plane, which cannot be assigned to another plane;
- sdeb, sfin: reference fiber point numbers, supports a line contained in the projection plane, if it is connected to the beam (GLOBAL or LOCAL options); sfin > sdeb.

If the projection plane is connected to the beam globally (GLOBAL option), it also contains a line parallel to the O_bZ_b axis, if it is an elevation view (ELEVATION option); the plan view plane (PLAN option) is orthogonal.

If O_bZ_b is vertical and ascending, a standing observer looks at the elevation with sdeb to his left and sfin to his right; the plan view is similar to a view from above, the observer conventionally looking in the opposite direction to that of O_bZ_b

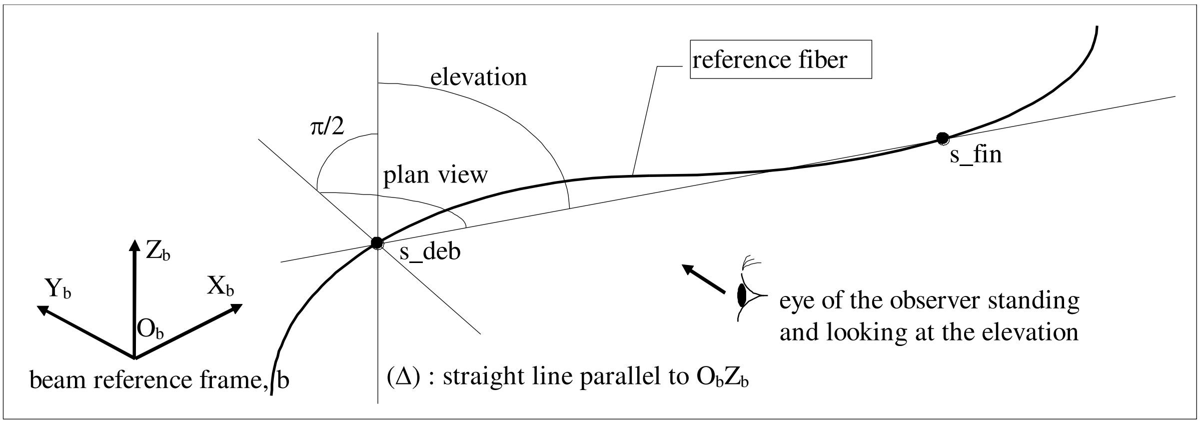


Figure 5.1 - Projection planes connected to the beam globally

If the projection plane is connected to the beam locally (LOCAL option), replace the line (Δ) (Figure 5.1) by the mean (Δ_1) of the generic o_iz_i axes at the points sdeb and sfin.

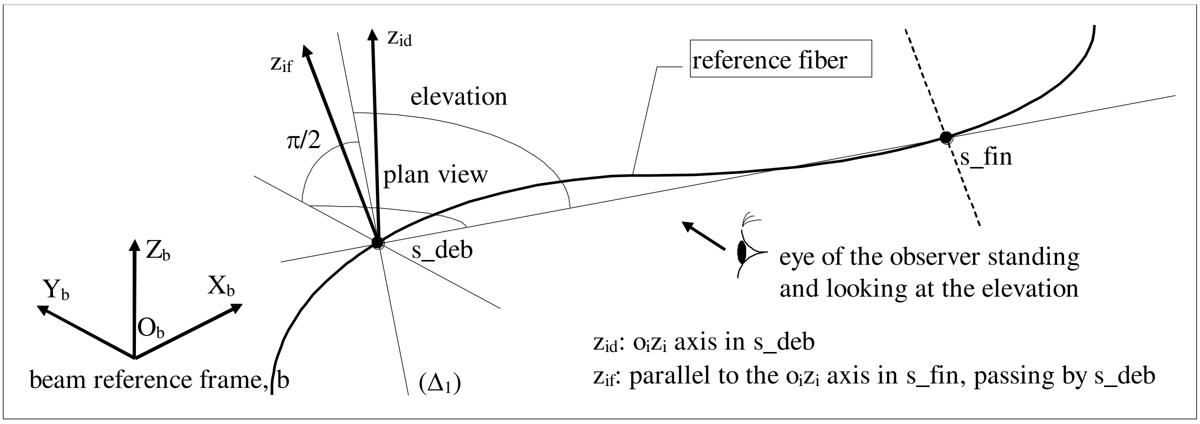


Figure 5.2 - Projection planes connected to the beam locally

- vnx, vny, vnz: components in the beam reference frame of the *normalized* vector $\vec{V}_n$ normal to the plane to be defined, if it is of any orientation (QUELCONQUE option), the direction indicates the direction of observation.

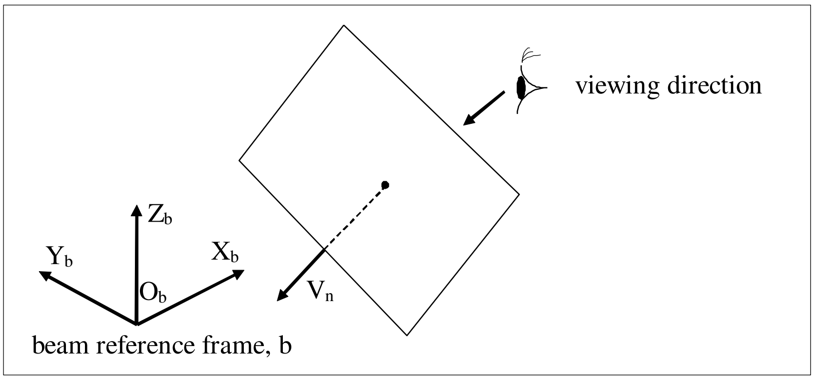


Figure 5.3 - Projection plane of any orientation

Functions

This command provides a projection plane to be used for formwork or cabling drawings.

Conditions of use

- The use of this command is mandatory if introducing at least one formwork or cabling drawing command, otherwise optional.
- All defined projection planes are not necessarily used (planes in reserve), but the validity of their existence is checked.

Methodological advice

- To obtain an approximation of the expanded view of a curved beam, it can be cut into segments, and each segment projected on a plane connected to the beam that is supported on the delimiting reference fiber points.
- For elevation views of cabling, choose planes orthogonal to those of the main cable beds, if they exist, to minimize proliferation.

Examples

```
$ drawings for elevations of a curved deck drawn span by span
PROJECTION ELEVATION GLOBALE  P_TR1    1   33 $ span 1
PROJECTION ELEVATION GLOBALE  P_TR2   33   80 $ span 2
PROJECTION ELEVATION GLOBALE  P_TR3   80  112 $ span 3

$ for the top view, plane ObXbYb, normal vector directed downwards
PROJECTION PLAN QUELCONQUE  P_TAB  0.0  0.0  -1.0
```

Related commands

FAMILLE CABLES ; DESSIN FIBRE ; DESSIN FAMILLE CABLES

5.4 - CABLE FAMILY

$$\text{FAMILLE} \quad \text{CABLES} \quad \begin{Bmatrix} \text{TOUS} \\ \text{LISTE} \\ \text{PARTICULE} \end{Bmatrix} \quad \text{fami} \quad [\text{psel}] \quad \text{titre\_famille}$$

$$[\left\langle \text{nom\_cable} \right\rangle_{\text{nb\_cables}}]$$

Parameters

- fami: cable family name, which cannot be assigned to another family;
- psel: selection of a particle (similar to a name), if cables selected by particle (PARTICULE option); all cables whose names contain this particle at least once will be retained;
- titre_famille: string identifying the cable family, will be reproduced on all drawings relating to it;
- nom_cable: name of a cable to be drawn, if cables selected by list (LISTE option); the number of names provided determines the value of the counter nb_cables, not explicitly defined.

Functions

This command is to select a group or family of cables to be drawn.

Conditions of use

- The use of this command is subject to the presence of at least one cable path in the definition of the beam by the GE1 module.
- The use of this command is mandatory if at least one cable drawing command is introduced, optional otherwise.
- It is not permissible to define several families of cables with the same list of names, in any order; the FAMILLE CABLES TOUS command can only appear once at most.
- All defined cable families are not necessarily subject to drawing (families in reserve), but the validity of their existence is checked.

Methodological advice

- In general, the elevation view of caissons with multiple cores are made by successive selections of cables of each core, to avoid overlapping of routes and cable names.
- The all type family is commonly used for plan views.

Examples

```
FAMILLE CABLES TOUS  FTC 'TOTALITE DES CABLES'

FAMILLE CABLES PARTICULE  AME_G  G  'CABLES DE L''AME GAUCHE'

FAMILLE CABLES LISTE  FGV  'SELECTION DE GAINES VIDES'
P1F1GV1  P1F1GV2
P1T1GV1  P1T1GV2
P1F2GV1  P1F2GV2
```

Related commands

PROJECTION ; DESSIN FAMILLE CABLES

5.5 – SECTION DRAWINGS

DESSIN SECTIONS [ECHELLES] nb_tron c_noeu c_cont c_cabl [echl echh] $\langle \text{sdeb} \quad \text{sfin} \rangle_{\text{nb\_tron}}$

Parameters

- nb_tron: number of beam elements, all of whose segments are drawn, positive.

A section drawing always includes its generic reference frame (raw drawing) or principal reference frame (fine drawing) and a representation of the geometry of its polygonal contours or its walls, depending on the method adopted for its definition, in the data of the GE1 module; other information is transferred to options.

- c_noeu: enter 1 to draw the polygonal contour section nodes (morphology), their eventual slabs, and their mean line, if known (fine drawing), 0 otherwise, or if the beam has no section of this type;
- c_cont: enter 1 to transfer the stress calculation points (normal, tangential and normal), 0 otherwise;
- c_cabl: enter 1 to transfer all the cable crossing points provided (raw drawing) or provided and determined by the GE3 module (fine drawing), 0 otherwise, or if the prestress is not defined;
- echl, echh: scale factors imposed in width and height; these values are provided only if the ECHELLES option is used, otherwise the scale factors of the ECHELLES STANDARD command are taken into account; see ECHELLES STANDARD command for their meaning;
- sdeb, sfin: reference fiber point numbers at the beginning and end of a beam segment; sfin ≥ sdeb; these values are equal for a segment reduced to a section; the sections must be disjoint and provided in ascending order of section numbers.

Functions

This command is to draw a series of sections grouped in continuous intervals, with transfer to options of certain dressing details.

Conditions of use

- ☐ The use of this command is optional.
- ☐ The presence of at least one section drawing command without scale specification mandates the ECHELLES STANDARD command.

Methodological advice

- ☐ In general, no scale distortion is applied to section drawings (equal scale factors in width and height).
- ☐ Do not simultaneously transfer morphology and cable crossing points (clarity).
- ☐ The points of calculation of stresses located at the height of the center of gravity (provided in the data of the GE1 module with a fictitious coordinate) are declared outside drawing frames by the GE2 module (raw drawing), and represented by the GE4 module in their positions calculated by the GE3 module (fine drawing).
- ☐ The continuous numbering of the cable crossing points, relative to each section, corresponds to the digital outputs of the GE1 module (raw drawing) or of the GE3 module (fine drawing).

Examples

```
ECHELLES STANDARD  50.0  50.0
$ drawing of four isolated sections with cable passage points
$                 n°_sections  c_nodes  c_cont  c_cable
DESSIN SECTIONS   4        0       0       1
5 5   12 12   14 14   33 33

$ drawing of an isolated section and all the sections of a half-beam
$ with the morphology (+top slab) (+mean line) and stress points
$                    n°_sections  c_nodes  c_cont  c_cable  scale   scale
DESSIN SECTIONS ECHELLES  2        1       1       0        25.0   25.0
1 1   7 33
```

Related commands

DESBRUT / DESFIN ; ECHELLES STANDARD

5.6 – FIBER DRAWING

DESSIN FIBRE [ECHELLES] plan sdeb sfin [echl echh]

Parameters

- plane: name of the projection plane used, defined by a PROJECTION command;
- sdeb, sfin: reference fiber point numbers at the beginning and end of the beam section for which formwork is being drawn; sfin > sdeb;
- echl, echh: scale factors imposed in width and height; these values are provided only if the ECHELLES option is used, otherwise the scale factors of the ECHELLES STANDARD command are taken into account; see ECHELLES STANDARD command for their meaning;

Functions

This command is to draw a projected formwork segment, with reference fiber (raw drawing), or mean fiber (fine drawing).

Following the dominant factor of the projection plane selected (plan or elevation), the formwork shell will be included calculated along the generic o_iy_i or o_iz_i axes of the sections, respectively.

Conditions of use

- ☐ The use of this command is optional.
- ☐ The presence of at least one formwork drawing command mandates defining at least one projection plane.
- ☐ The presence of at least one formwork drawing command without specification of scale mandates the ECHELLES STANDARD command.

Examples

```
$ elevation views following the curve
ECHELLES STANDARD  100.0  50.0
DESSIN FIBRE  P_TR1   1   33 $ span 1
DESSIN FIBRE  P_TR2  33   80 $ span 2
DESSIN FIBRE  P_TR3  80  112 $ span 3

$ view in global plane
DESSIN FIBRE ECHELLES  P_TAB  1  112  100.0  100.0
```

Related commands

DESBRUT / DESFIN ; ECHELLES STANDARD ; PROJECTION

5.7 - CABLE FAMILY DRAWINGS

DESSIN FAMILLE CABLES [ECHELLES] fami plan sdeb sfin rcab [echl echh]

Parameters

- fami: family name of the cables to be drawn, defined by a FAMILLE CABLES command;
- plane: name of the projection plane used, defined by a PROJECTION command;
- sdeb, sfin: reference fiber point numbers at the beginning and end of the beam section for which cabling is being drawn; sfin > sdeb;
- rcab: enter 1 if the names of the cables should be transferred, preferentially to their anchors, 0 otherwise;
- echl, echh: scale factors imposed in width and height; these values are provided only if the ECHELLES option is used, otherwise the scale factors of the ECHELLES STANDARD command are taken into account; see ECHELLES STANDARD command for their meaning;

Functions

This command is to draw a projected beam section, with some of its cables, in the form of broken lines with arrows showing the tangent vectors (raw drawing), or smoothed curves tabulated at 0.25 m spacings (fine drawing); the transfer of cable names is optional.

Following the dominant factor of the projection plane selected (plan or elevation), the formwork shell will be included calculated along the generic o_iy_i or o_iz_i axes of the sections, respectively.

Conditions of use

- ☐ The use of this command is optional.
- ☐ The use of this command is subject to the presence of at least one cable path in the definition of the beam by the GE1 module.
- ☐ The presence of at least one cable drawing command mandates defining a projection plane and at least one cable family.
- ☐ The presence of at least one cable drawing command without scale specification mandates the ECHELLES STANDARD command.
- ☐ The cable family segment to be projected must not be empty.

Methodological advice

- Avoid transferring cable names if the views are too "extensive" because the size of their characters depends on the selected equipment, it is not dependent on scale factors used; overlapping names are not tolerated (collision management occurs through incremental displacement of the names along the projected cable routes), the number of messages indicating the inability to transfer some names may be high.
- If the segment to be drawn begins in the first section of the beam, and/or terminates at its final section, the portions of the corresponding cables, possibly protruding from the longitudinal region of the beam, are not clipped.

Examples

```
ECHELLES STANDARD 100.0 20.0
$ elevation views following the curve, left core cables
DESSIN FAMILLE CABLES  AME_G  P_TR1   1  33   1 $ span 1
DESSIN FAMILLE CABLES  AME_G  P_TR2  33  80   1 $ span 2
DESSIN FAMILLE CABLES  AME_G  P_TR3  80 112   1 $ span 3

$ global plane view, all cables
DESSIN FAMILLE CABLES ECHELLES  FTC  P_TAB  1  112  1  100.0  100.0
```

Related commands

DESBRUT / DESFIN ; ECHELLES STANDARD ; PROJECTION ; FAMILLE CABLES

5.8 - END

FIN

Functions

This order marks the end of a beam data drawing data group.

Conditions of use

- Must be the last line of the command file.

Examples

```
DESBRUT  100
............
............
FIN
```

Related commands

DESBRUT / DESFIN

Chapter 6

Definition

Introduction

This chapter describes the data needed to define the general mechanical model, including integration parameters for (spatial) beams, non-beam elements with their characteristics, and certain additional elements (articulation-types, support-types, movable equipment).

Data

One file should be produced for the model, its size is not limited. In particular, the number of commands to be introduced is not limited, as well as the possible data volume associated with a particular command. For example, we can introduce as many section-types and materials as required, and as many nodes and elements as necessary (dynamically allocated memory).

The only existing limit concerns the number of beams, which must not exceed 9999.

Some "excess" commands are accepted (materials and section-types for non-beam elements in reserve, unused); however, no excess data is tolerated in a command (nodes or surplus elements with respect to numbers declared, etc.).

The data, presented in a logical and hierarchical order for their description, can be introduced in any order (except for start and end of file commands), forward references are possible (current elements receiving material assignments and section-types not yet defined, elements connecting nodes not yet defined, etc.).

The order of entry, however, determines the internal storage order of certain entities and their order of appearance in the numerical results (materials, section-types, etc.).

With no end delimiter, each command in the PH1 module must begin on a new line and the first line break of the wording is to be observed. However, cutting data that follows a command header into lines, is unrestricted.

Analysis mode

The PH1 module performs compilation type data analysis (coupled with a control), by hierarchical levels and successive scans of the entire file, from the broadest to the finest, stopping at the level where the first error occurs.

The number of possible errors that can be detected is not limited for a level, but the detection of an error of a certain level of severity, for a command, may prevent the search for other errors on the same level.

For example, meeting an incorrect keyword in a section prevents analysis of the content and the detection of a lexically incorrect integer or real value in an (alpha)numerical list will stop its analysis.

Processing sequence

If the PH1 module has detected no errors from the lowest level, the model is declared to be correct and recorded in a homogeneous form in the database (nodes and elements combined in a fixed order, beam cables grouped together, beam materials and non-beam elements merged, etc.).

Editing

The PH1 module outputs the status of the recorded model reduced to its main tables (element-types, beams, nodes, elements, etc.).

The volume of these outputs is not scalable.

Contents

6.1- MODEL

MODELE

Functions

This command marks the start of the model data.

Conditions for use

- ☐ Must be the first line of the command file.

Examples

```
MODELE
......
......
FIN
```

Related commands

TITRE ; FIN

6.2 - TITLE

TITRE   titre_modele

Parameters

- titre_poutre: character string

Functions

The first TITRE command met holds the main title of the model, which is at the top of each page of the digital outputs (of the PH1 module) and each drawing (of the PH2 module), thereof.

The main title is also found at the start of the results of the PH1 module, and at the start and the top of each page of the results of the PH3 module, until it has been updated with a TITRE command specific to this module.

The content of all TITRE commands (including the first) is also reproduced at the start of the results of the PH1 module.

Conditions for use

☐ Introduce at least one command of this type.

Examples

```
TITRE  '*** VIADUC D''ACCES B, MODELE ***'
TITRE  '-------------------------------'
```

Related commands

MODELE

6.3 - MATERIAL

MATERIAU nom_materiau nb_parametres

loi_fluage loi_module loi_retrait

module_reference coef_poisson poids_volumique coef_dilatation

$\langle \text{para\_supp} \rangle_{\text{nb\_parametres}}$

Parameters

- nom_materiau: name of the material, which cannot be assigned to another material of the model (of beams and non-beam elements);
- nb_parametres: number of additional parameters associated with the rheological behavior laws, indicate 3;
- loi_fluage: creep law number;
- loi_module: instantaneous Young's modulus variation law number;
- loi_retrait: shrinkage law number;
- module_reference: Young's modulus used in the absence of fine rheological calculation;
- coef_poisson: Poisson's ratio, must be greater than -1.0;
- poids_volumique: specific weight;
- coef_dilatation : thermal expansion coefficient;
- para_supp: additional parameter whose content depends on the rheological behavior laws chosen.

See Annex B for the meaning of the numbers of rheological behavior laws and their additional parameters.

When an additional parameter is not used, enter a readable fictitious value (1.0).

The parameters module_reference, poids_volumique and coef_dilatation must be greater than 1.0E-8.

Functions

This command provides the basic material for non-beam elements, characterized by physical parameters and behavioral laws over time, for certain phenomena, set by the regulations.

These behavioral laws and their associated parameters are used by the PH3 module in CALCUL RHÉOLOGIQUE FIN mode.

Conditions of use

- ☐ Do not define material if the model has no current element.
- ☐ If the model has at least one current element, provide at least one material; the number of materials can then be arbitrary, some may remain in reserve and not assigned.

Methodological advice

- Beam materials being "visible" only at the the level of their constituent data (GE1 module), are duplicated from one beam to another by changing their names, if they have the same properties.
- In the same vein, if non-beam elements and beam elements are made of identical materials, it is necessary to redefine them in the model data with different names.
- Assign a very low specific weight to the constituent materials of "rigid" elements defined as current elements with strongly enhanced characteristics. If their own weight cannot be taken into account upon activation, their initial mass is calculated by the DYN module.
- For the choice of the module_reference parameter, see the methodological guidance for the MATERIAU command of the GE1 module (Chapter 3).

Examples

```
$ BPEL-compliant concrete
MATERIAU  PILBETON  3
3                     $ creep law
14                    $ variation law of the instantaneous Young’s modulus
2                     $ retraction law
3.60E06               $ reference Young’s modulus
0.20                  $ Poisson coefficient
2.50                  $ unit weight
1.0E-05               $ thermal expansion coefficient
3.60E06               $ instantaneous modulus after 28 days
1.0                   $ unused parameter
0.95                  $ parameter KS of the regulation
```

Related commands

SECTION TYPE ; ELEMENTS COURANTS

6.4 – SECTION TYPE

SECTION TYPE nom_section nb_sigma nb_tau

surface rigid_fy rigid_fz inertie_mx inertie_my inertie_mz

yp_centre zp_centre e_moyenne

[$\langle$sigma_fx sigma_my sigma_mz$\rangle_{\text{nb_sigma}}$]

[$\langle$tau_fx tau_fy tau_fz tau_mx tau_my tau_mz

sigmac_fx sigmac_my sigmac_mz$\rangle_{\text{nb_tau}}$]

Parameters

- nom_section: section-type name, which cannot be assigned to another section-type;
- nb_sigma: number of normal stress calculation points, positive or zero;
- nb_tau: number of "generalized" tangential and concomitant normal stress calculation points, positive or zero;
- surface: surface;
- rigid_fy, rigid_fz: shear stiffness according to the principal axes Gy and Gz; in the absence of a precise knowledge of these parameters, we can introduce:
  - shear force reduced sections;
  - zero values indicating that the deformation due to shear force is not taken into account;
- inertie_mx: pure torsion inertia (St Venant constant);
- inertie_my, inertie_mz: principal bending inertia according to the principal axes Gy and Gz;
- yp_centre, zp_centre: coordinates of the center of torsion in the *principal reference frame;*
- e_moyenne: surface/outer perimeter ratio, used by PH3, in CALCUL RHÉOLOGIQUE FIN mode, in certain materials shrinkage and creep laws (see reference documents listed in Annex B).
- sigma_fx, sigma_my, sigma_mz: weighting coefficients of the forces Fx, My and Mz for the calculation of the normal stresses at a point; to be provided if nb_sigma is positive;
- tau_fx, tau_fy, tau_fz, tau_mx, tau_my, tau_mz: weighting coefficients of the forces Fx, Fy, Fz, Mx, My and Mz, for calculating the generalized tangential stresses at a point; to be provided if nb_tau is positive;
- sigmac_fx, sigmac_my, sigmac_mz: weighting coefficients of the forces Fx, My and Mz for the calculation of concomitant normal stresses; to be provided if nb_tau is positive.

Functions

This command defines a section-type to be assigned to certain current non-beam elements, by its mechanical properties, and any coefficients permitting the calculation of stresses (normal, generalized tangential and concomitant normal stresses), in certain points, from the forces applied in the principal reference frame.

Conditions for use

- Do not define a section-type if the model has no current element.
- If the model has at least one current element, provide at least one section-type; the number of section-types can then be arbitrary, some may remain in reserve and not assigned.
- The first six basic characteristics (surface to inertie_mz) must be positive or zero, their sum must remain positive; the parameter e_moyenne must be greater than 0.0001.
- Among the nine basic characteristics (surface to e_moyenne), only surface and e_moyenne must be different from 0.0 for a bi-articulated element (recognition criterion).
- Non bi-articulated current elements are declared "standard".

Methodological advice

- Avoid mixing "rigid" elements defined as current elements with strongly enhanced characteristics and elements introduced by the ELEMENTS RIGIDES command.

Examples

In the section below, tangential stresses are calculated with the simplified theory, and bending forces have no influence on them.

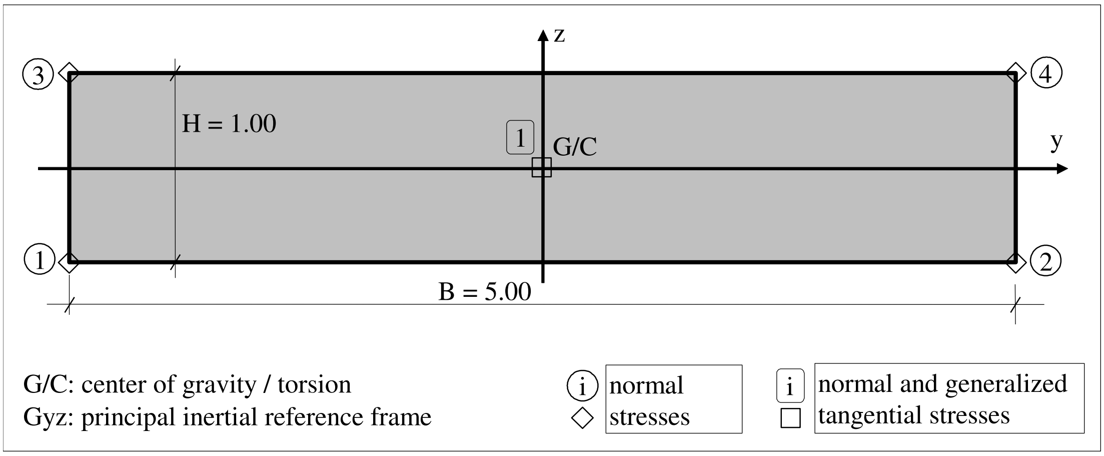


Figure 6.1 - Rectangular section

```
$ parameterized rectangular section, for standard elements
$ to adapt to other dimensions, change B, H and K2
<B   = 5.0>
<H   = 1.0>
<K2  = 0.291>        $ for constant St Venant and B/H ratio=5.0
<S   = (B*H)>
<R_Y = (S*5.0/6.0)>
<R_Z = (R_Y)>
<I_X = (K2*B*H**3.0)>
<I_Y = (B*H**3.0/12.0)>
<I_Z = (H*B**3.0/12.0)>
<E_M = (0.5*S/(B+H))>
<M_Y = (B*H**2.0/8.0)>
<M_Z = (H*B**2.0/8.0)>
```

```
SECTION TYPE  RECT_01  4  1
(S)   $ surface area
(R_Y) $ rigid_fy
(R_Z) $ rigid_fz
(I_X) $ inertia_mx
(I_Y) $ inertia_my
(I_Z) $ inertia_mz
0.0   $ yp_center
0.0   $ zp_center
(E_M) $ e_mean
$ calculation points of the normal stresses
$ sigma_fx   sigma_my       sigma_mz
(1.0/S)      ( 0.5*H/I_Y)   (-0.5*B/I_Z) $ point 1
(1.0/S)      ( 0.5*H/I_Y)   ( 0.5*B/I_Z) $ point 2
(1.0/S)      (-0.5*H/I_Y)   (-0.5*B/I_Z) $ point 3
(1.0/S)      (-0.5*H/I_Y)   ( 0.5*B/I_Z) $ point 4
$ calculation points of the generalized tangential and normal stresses
$ tau_fx   tau_fy         tau_fz         tau_mx    tau_my    tau_mz
  0.0      (M_Z/(I_Z*H))  (M_Y/(I_Y*B))  0.0       0.0       0.0
$ sigmac_fx   sigmac_my   sigmac_mz
  (1.0/S)     0.0         0.0

$ section for bi-articular elements
$ no stress calculation points
SECTION TYPE  HAUB_9  0  0
$ surface           e_mean
0.007854   7*0.0   0.05
```

Related commands

MATERIAU ; ELEMENTS COURANTS

6.5 - ARTICULATION

$$\text{ARTICULATION} \quad \begin{Bmatrix} \text{RIGIDITE} \\ \text{PARFAITE} \end{Bmatrix} \quad \text{nom\_arti}$$

$$\begin{Bmatrix} \text{rigid\_fx} & \text{rigid\_fy} & \text{rigid\_fz} & \text{rigid\_mx} & \text{rigid\_my} & \text{rigid\_mz} & \\ \text{bloq\_fx} & \text{bloq\_fy} & \text{bloq\_fz} & \text{bloq\_mx} & \text{bloq\_my} & \text{bloq\_mz} & [\text{c\_pond}] \end{Bmatrix}$$

Parameters

- nom_arti: articulation--type name, which cannot be assigned to another articulation--type;
- rigid_fx, rigid_fy, rigid_fz, rigid_mx, rigid_my, rigid_mz: stiffness coefficients (positive or zero) connecting the forces Fx, Fy, Fz, Mx, My and Mz, to the displacements u_x, u_y, u_z, θ_x, θ_y and θ_z, in the articulation--type local reference frame, if it is defined by its diagonal stiffness matrix (RIGIDITE option);
- bloq_fx, bloq_fy, bloq_fz, bloq_mx, bloq_my, bloq_mz: coefficients connecting forces and displacements above, for a perfect articulation-type (PARFAITE option); indicate 0 if the corresponding displacement is free, 1 if it is blocked on a weighted basis;
- c_pond: common weighting coefficient for blocked displacements of a perfect articulation-type; this value will be determined by the PH3 module if it is not provided; indicate 1.0E7 for "perfect" blocking in the current cases (absolute coefficient).

Functions

This command defines an ad hoc articulation-type, by its stiffness matrix, or degrees of freedom blocked on a weighted basis.

Conditions for use

- Do not define an articulation-type if the model contains no articulation elements.
- If the model has at least one articulation, define at least one articulation-type; articulation-types, placed or replaced by the PH3 module during construction, are not subject to initial assignment to model elements.

Methodological advice

- Reduce or do not define the weighting coefficients defined for perfect articulation-types that cause type 3 instabilities during construction.

Examples

```
ARTICULATION RIGIDITE  ARTI_1
$ rigid_fx  rigid_fy  rigid_fz  rigid_mx  rigid_my  rigid_mz
  1.0E6     1.0E4     1.0E4     1.0E3     1.0E2     1.0E3

ARTICULATION PARFAITE  ARTI_BLO
$ bloq_fx  bloq_fy  bloq_fz  bloq_mx  bloq_my  bloq_mz
  1        1        1        1        1        1

ARTICULATION PARFAITE  P_NEOP1
$ bloq_fx  bloq_fy  bloq_fz  bloq_mx  bloq_my  bloq_mz  c_pond
  0        0        1        0        0        0        1.0E5
```

Related commands

ELEMENTS ARTICULATIONS

6.6 - SUPPORT

$$\text{SUPPORT} \quad \begin{Bmatrix} \text{RIGIDITE} \\ \text{SOUPLESSE} \\ \text{PARFAIT} \end{Bmatrix} \quad \text{nom\_appui}$$

$$\begin{Bmatrix} \langle \text{m\_rigidite} \rangle_{21} \\ \langle \text{m\_souplesse} \rangle_{21} \\ \text{bloq\_fx} \quad \text{bloq\_fy} \quad \text{bloq\_fz} \quad \text{bloq\_mx} \quad \text{bloq\_my} \quad \text{bloq\_mz} \quad [\text{c\_pond}] \end{Bmatrix}$$

Parameters

- nom_appui: support--type, which cannot be assigned to another support--type;
- m_rigidite: stiffness coefficient connecting a force Fx, Fy, Fz, Mx, My or Mz, to a displacement u_x, u_y, u_z, θ_x, θ_y or θ_z, in the support--type local reference frame, if it is defined by its full stiffness matrix (RIGIDITE option); the 21 coefficients represent the upper triangular matrix described line by line;

$$\begin{bmatrix} Fx \\ Fy \\ Fz \\ Mx \\ My \\ Mz \end{bmatrix} = \begin{bmatrix} 1 & 2 & 3 & 4 & 5 & 6 \\ . & 7 & 8 & 9 & 10 & 11 \\ . & . & 12 & 13 & 14 & 15 \\ . & . & . & 16 & 17 & 18 \\ . & . & . & . & 19 & 20 \\ . & . & . & . & . & 21 \end{bmatrix} \times \begin{bmatrix} u_x \\ u_y \\ u_z \\ \theta_x \\ \theta_y \\ \theta_z \end{bmatrix} \qquad (6.1)$$

- m_souplesse : flexibility coefficient connecting a displacement u_x, u_y, u_z, θ_x, θ_y or θ_z, to a force Fx, Fy, Fz, Mx, My or Mz, in the support--type local reference frame, if it is defined by its full flexibility matrix (SOUPLESSE option); the 21 coefficients represent the upper triangular matrix described line by line; negative or zero terms are not allowed on the diagonal; terms greater than the maximum diagonal term divided by 1.0E7 are considered real flexibility coefficients; terms less than or equal to this value are considered coefficients to be ruled out "provisionally", together with their corresponding rows and columns (necessarily zero terms), before inversion of the matrix, and then to be replaced with their inverses "boxed" by zeros, on the diagonal of the inverse matrix;

$$\begin{bmatrix} u_x \\ u_y \\ u_z \\ \theta_x \\ \theta_y \\ \theta_z \end{bmatrix} = \begin{bmatrix} 1 & 2 & 3 & 4 & 5 & 6 \\ . & 7 & 8 & 9 & 10 & 11 \\ . & . & 12 & 13 & 14 & 15 \\ . & . & . & 16 & 17 & 18 \\ . & . & . & . & 19 & 20 \\ . & . & . & . & . & 21 \end{bmatrix} \times \begin{bmatrix} Fx \\ Fy \\ Fz \\ Mx \\ My \\ Mz \end{bmatrix} \qquad (6.2)$$

- bloq_fx, bloq_fy, bloq_fz, bloq_mx, bloq_my, bloq_mz: coefficients connecting forces and displacements above, for a perfect support--type, defined by its diagonal stiffness matrix (PARFAIT option); indicate 0 if the corresponding displacement is free, 1 if it is blocked in a weighted manner;

- c_pond: common weighting coefficient for blocked displacements of a perfect support-type; this value will be determined by the PH3 module if it is not provided; indicate 1.0E5 for "perfect" blocking in the current cases (relative coefficient).

Functions

This command defines a support--type by its full stiffness or flexibility matrix, or degrees of freedom blocked on a weighted basis.

Conditions for use

- ❐ Define at least one support-type.

Methodological advice

- ❐ Reduce or do not define the weighting coefficients defined for perfect support--types that cause type 3 instabilities during construction.

Examples

```
$ south pile, type 1 support (rigidity matrix, upper triangular)
APPUI RIGIDITE  FPISU1
0.0520E+6   0.0         0.0          0.0         -0.1916E+6   0.0
            0.0521E+6   0.0          0.1941E+6    0.0         0.0
                        0.5319E+6    0.0          0.0         0.0
                                     9.8144E+6    0.0         0.0
                                                  2.9782E+6   0.0
                                                              1.0204E+6

$ flexibility matrix that is the inverse of the above matrix, the maximum
term of $ the diagonal is 0.252057E-4, no term of the latter being inferior
to $ 0.252057E-4/1.0E+7, the matrix will be globally inverted for its
$ integration in the calculations, without extracting any line or column
APPUI SOUPLESSE  FPISU1
0.252057E-4 0.0         0.0          0.0          0.162159E-5 0.0
            0.207205E-4 0.0         -0.409792E-6  0.0         0.0
                        0.188005E-5  0.0          0.0         0.0
                                     0.109996E-6  0.0         0.0
                                                  0.440097E-6 0.0
                                                              0.980008E-6

APPUI PARFAIT  PILE_NEO
$ bloq_fx  bloq_fy  bloq_fz  bloq_mx  bloq_my  bloq_mz
  0        0        1        1        0        0

APPUI PARFAIT  PILE_NEO
$ bloq_fx  bloq_fy  bloq_fz  bloq_mx  bloq_my  bloq_mz  c_pond
  0        0        1        1        0        0        1.0E5
```

Related commands

NOEUDS FIBRE POUTRE ; NOEUDS REPERES POUTRE

NOEUDS REPERE GLOBAL

6.7 – FORM TRAVELER

EQUIPAGE MOBILE nom_equipage nb_points

x_poids y_poids poids

$\langle \text{x\_point} \quad \text{y\_point} \rangle_{\text{nb\_points}}$

Parameters

- nom_equipage: name of the form traveler that cannot be assigned to another form traveler;
- nb_points: number of load application points, at least 3;
- x_poids, y_poids: coordinates of the crossing point of the line of action of the total weight of the form traveler in the generic reference frame $o_ix_iy_i$ attached when it is placed along a beam (see Chapter 8, PLACER EQUIPAGES command);
- weight: total weight;
- x_point, y_point: coordinates in the generic reference frame $o_ix_iy_i$ of the load M_j application point; it is verified that these points are not all aligned (overall stability).

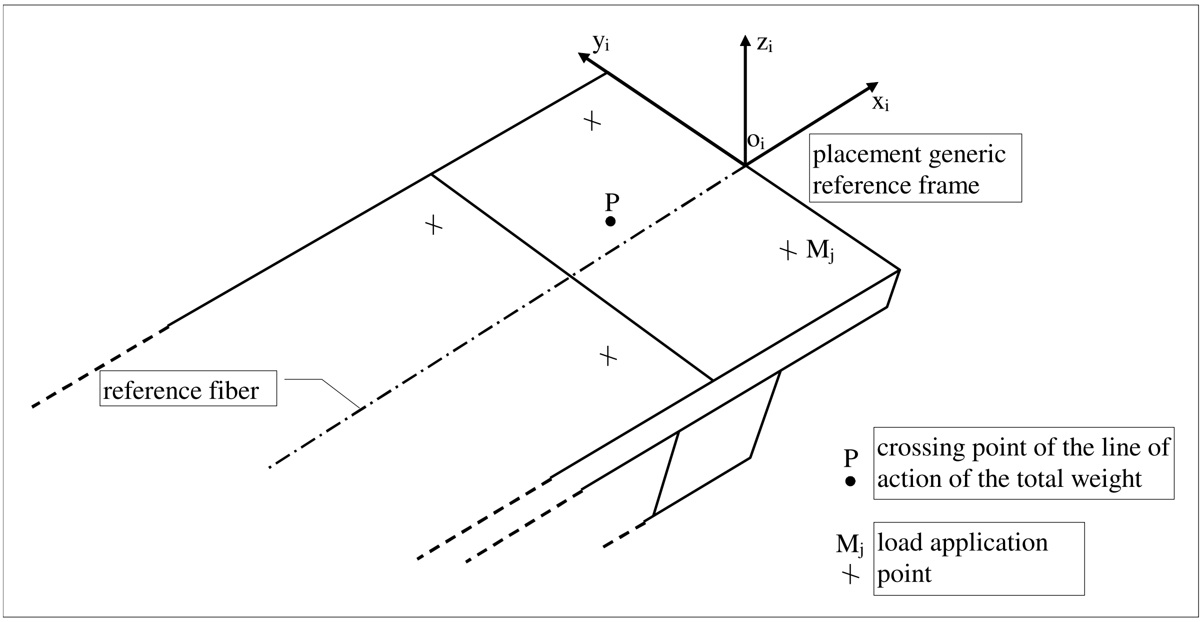


Figure 6.2 - Form traveler placed on beam in construction

Functions

This command provides the characteristics of a form traveler for the implementation of beams.

The presence of form travelers is useful for the placement of suspended beam elements by the PH3 module (see Chapter 8, PLACER ELEMENTS command, SUSPENDUS option).

Conditions of use

- Do not define a form traveler if the model does not include a beam.
- If the model has at least one beam, the use of this command is accepted, but not mandatory.

Methodological advice

- A form traveler cannot be placed simultaneously at different locations during construction, it must be duplicated as many times as necessary, if appropriate, changing its name.

Examples

```
EQUIPAGE MOBILE  COUL_VO1  4
$  x_weight  y_weight  weight
   -2.25     0.00      60.0
$ x_point   y_point      x_point   y_point
  -3.50     4.75         -3.50     -4.75
  -1.00     4.75         -1.00     -4.75
```

Related commands

NOEUDS FIBRE POUTRE ; ELEMENTS POUTRE

6.8 - BEAM POSITION

POSITION POUTRE no_poutre

x_trans y_trans z_trans θ_1 θ_2 θ_3

Parameters

- no_poutre: number of a beam to be integrated in the model, by positioning a recorded beam;
- x_trans, y_trans, z_trans: coordinates of the origin of its definition reference frame in the global reference frame;
- θ_1, θ_2, θ_3: standard triple rotation (see Figure 1.2) orienting its definition reference frame, relative to the global reference frame.

Functions

This command provides the positioning data of a beam belonging to the model and not defined in the global reference frame.

Conditions for use

- ❐ Do not introduce this type of command if the model does not include a beam.
- ❐ If the model has at least one beam, provide a command of this type for each beam belonging to it, and not defined in the global reference frame.
- ❐ The POSITION POUTRE command is optional for all beams belonging to the model and defined in the global reference frame (to which zero values of x_trans, y_trans, z_trans, θ_1, θ_2 and θ_3 are assigned by default).
- ❐ The no_poutre number must be part of the model's reference beam list, set by the pair of commands NOEUDS FIBRE POUTRE and ELEMENTS POUTRE; it cannot be used simultaneously for a beam defined in the global reference frame and positioned implicitly, or for a beam obtained by copying (second parameter of the COPIE POUTRE command).

Methodological advice

- ❐ All beams whose average development direction is parallel to the axis OZ of the global reference frame, or is close to it, must be described in an adequate local reference frame and positioned in the global reference frame.

Examples

The model has two beams numbered 10 and 20, previously recorded under these numbers; the beam 10 is defined directly in the global reference frame, and occupies a part of its OX axis, and therefore is not subject to positioning; the beam 20 is described in a local reference frame, and occupies a part of its $O_{p20}X_{p20}$ axis, derived from the global reference frame by a double translation and a single rotation.

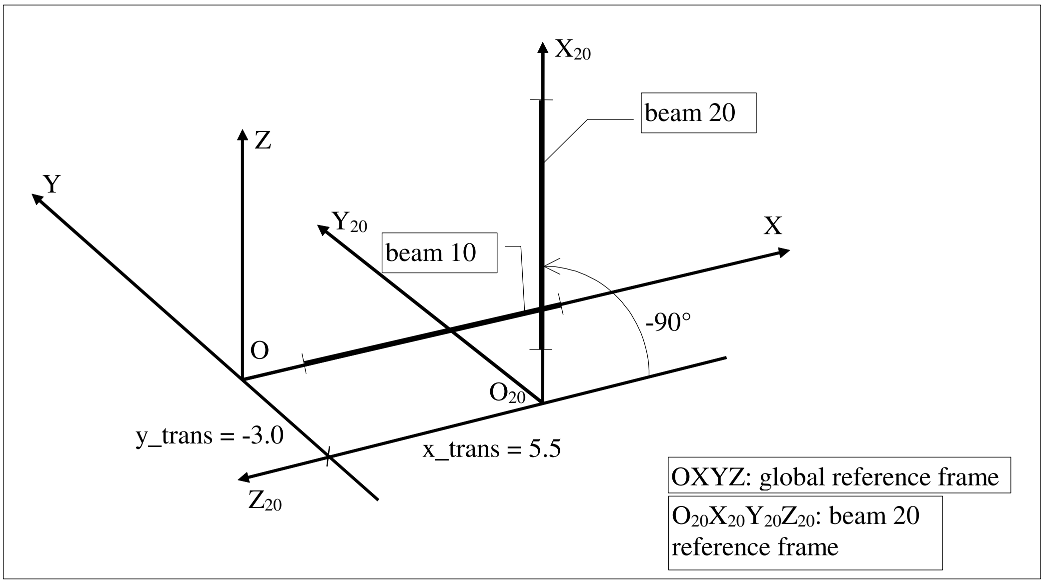


Figure 6.3 - positioned (or copied) beam

```
$ this sequence of commands determines the list of reference beams
NOEUDS FIBRE POUTRE  10  (..)
.............................
NOEUDS FIBRE POUTRE  20  (..)
.............................
ELEMENTS POUTRE  10
...................
ELEMENTS POUTRE  20
...................
$ positioning of beam 20 by translation and rotation of its coordinates
POSITION POUTRE  20
$ x_trans  y_trans  z_trans  teta1  teta2  teta3
  5.5      -3.0     0.0      0.0    -90.0  0.0
```

Related commands

COPIE POUTRE ; NOEUDS FIBRE POUTRE ; NOEUDS REPERES POUTRE

ELEMENTS POUTRE

6.9 - BEAM COPY

COPIE POUTRE n1_poutre n2_poutre

x_trans y_trans z_trans θ_1 θ_2 θ_3 part_sel part_sub

Parameters

- n1_poutre: recorded beam number;
- n2_poutre: beam number to be integrated into the model, by copying the n1_poutre beam; n2_poutre ≠ n1_poutre;
- x_trans, y_trans, z_trans: coordinates of the origin of its definition reference frame in the global reference frame;
- θ_1, θ_2, θ_3: standard triple rotation (see Figure 1.2) orienting its definition reference frame, relative to the global reference frame;
- part_sel: selection particle (similar to a name), necessarily present in the names of materials, cable-types and cables of the n1_poutre beam;
- part_sub: particle (similar to a name), which will replace the first occurrences of part_sel to generate the names of materials, cable-types and cables of the n2_poutre beam; substitution must not cause length overhangs, nor duplicate names, in each category, relative to the full model (non-beam materials included).

Functions

This command provides beam repositioning data, to be integrated into the model by copying an "original" beam also belonging to the model.

Copying is of interest for the beam formwork, any cabling, and its constituent materials.

This technique allows us not define redundant beams in the GE1 module, to within a translation and/or a rotation.

The main label of a beam obtained by copying is that of the copied beam, prefixed by n2_poutre, in its first four characters; recording dates are also included from those of the copied beam.

Conditions for use

- Do not introduce this type of command if the model does not include a beam.
- If the model has at least one beam, provide a command of this type for each beam belonging to it, and obtained by copying.
- The n1_poutre number must be part of the model's reference beam list, set by the pair of commands NOEUDS FIBRE POUTRE and ELEMENTS POUTRE; it must correspond to an original beam defined in the global reference frame or positioned. It is therefore not permitted to copy a beam that is recorded but does not belong to the model, or to copy a beam obtained by copying (cascade copying).
- The n2_poutre number must also be part of the model's reference beam list; it should not correspond to an original beam defined in the global reference frame or positioned, nor to a beam recorded in the database (in reserve or referenced).

Methodological advice

- Please note that translation and triple rotation do not operate relative to the actual position occupied by n1_poutre if it is the subject of the positioning, but in "absolute" fashion, as if the beam n2_poutre were positioned.

Examples

The model has two identical beams numbered 10 and 20, beam 10 is previously recorded under this number; it is defined in the global reference frame directly and therefore is not subject to positioning; beam 20 is derived from beam 10 by copying, translation and rotation of its reference frame (Figure 6.3 can also be applied to this case).

It is verified that beam 20 is not recorded in the database.

```
$ this sequence of commands determines the list of reference beams
NOEUDS FIBRE POUTRE  10  (..)
.............................
NOEUDS FIBRE POUTRE  20  (..)
.............................
ELEMENTS POUTRE  10
...................
ELEMENTS POUTRE  20
...................
$ repositioning of beam 10 to obtain beam 20 by copying
COPIE POUTRE  10  20
$ x_trans  y_trans  z_trans  teta1   teta2   teta3
  5.5      -3.0     0.0      0.0     -90.0    0.0
$ part_sel  part_sub
  P10       P20
```

Related commands

POSITION POUTRE ; NOEUDS FIBRE POUTRE ; NOEUDS REPERES POUTRE

ELEMENTS POUTRE

6.10 - BEAM FIBER NODES

NOEUDS FIBRE POUTRE no_poutre [nb_noeuds]

$$\left\{\begin{array}{l} \text{noeud\_depart} \quad \text{pas\_n} \\ \left\langle \text{nb\_noeuds\_s} \quad \left\langle \text{no\_noeud} \right\rangle_{\text{nb\_noeuds\_s}} \right\rangle_{\text{nb\_points\_f}} \end{array}\right\}$$

Parameters

- no_poutre: number of a beam belonging to the model and obtained, either by direct recording followed by eventual positioning, or by copying and repositioning an existing beam;
- nb_noeuds: number of geometric nodes created by the introduction of the beam, at its mean fiber:
  - do not code this value if each section of the beam generates a single node; it is then assumed that these nodes are numbered by the PH1 module, in arithmetic progression, case 1;
  - indicate a value greater than or equal to the number of reference (or mean) fiber points of the beam (nb_points_f), if all node numbers are provided and/or if at least one mean fiber point is indicated more than once, case 2;
- noeud_depart, pas_n: starting value (positive) and node numbering step (positive or negative) in case 1;
- nb_noeuds_s, no_noeud: number of nodes generated by a beam section (greater than or equal to 1) and node number, in case 2.

In both cases, all beam nodes are listed in ascending order of the section numbers (direction of travel). Whether generated or provided, the node numbers must be unique (for the full model), positive and less than 1000_000.

In case 2, each point of the mean fiber must be described at least once, and it is verified that $\sum_{1}^{\text{nb\_points\_f}} \text{nb\_noeuds\_s} = \text{nb\_noeuds}$.

Functions

This command provides the node numbers belonging to the mean fiber of a beam in the model, and the way to re-position them.

Conditions for use

- Do not introduce this type of command if the model does not include a beam.
- If the model has at least one beam, provide a command of this type for each beam belonging to it (present directly or obtained by copying).
- NOEUDS FIBRE POUTRE and ELEMENTS POUTRE command groups (in equal numbers) determine the number of beams belonging to the model, and the reference list of their numbers (identical, in any order, from one group to another).

Methodological advice

- To add an internal articulation to a beam, point twice on the corresponding node.
- To introduce an articulation, at a beam mean fiber level, which does not break its continuity, the corresponding mean fiber node may be duplicated or a copy defined, as a non-beam node.

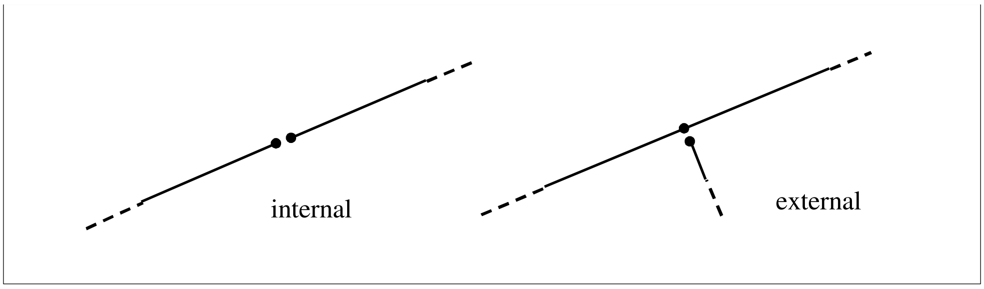


Figure 6.4 - Beam, articulations at a mean fiber level

Examples

Beam number 20 comprises 5 sections; in case 1, each section produces only one node (very common); their numbers are in arithmetic progression.

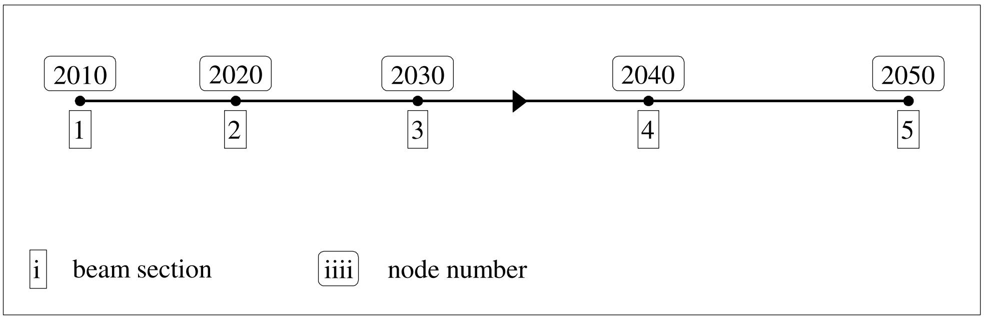


Figure 6.5 - Beam, mean fiber nodes, case 1

```
NOEUDS FIBRE POUTRE  20
$ start node  pitch_n
  2010           10
```

In case 2, each section produces a node, except for the third which is duplicated; all node numbers are provided.

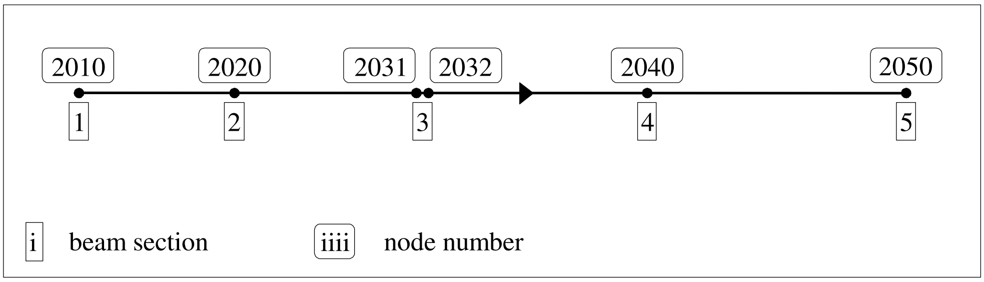


Figure 6.6 - Beam, mean fiber nodes, case 2

```
NOEUDS FIBRE POUTRE  20  6
$ n°_nodes_s  no node  no node     n°_nodes_s    no node  no node
  1             2010                 1             2020
  2             2031      2032       1             2040
  1             2050
```

Related commands

APPUI ; EQUIPAGE MOBILE ; POSITION POUTRE ; COPIE POUTRE

NOEUDS REPERES POUTRE ; NOEUDS REPERE GLOBAL

ELEMENTS POUTRE ; ELEMENTS COURANTS ; ELEMENTS RIGIDES

ELEMENTS ARTICULATIONS ; EXCENTREMENTS

6.11 - BEAM REFERENCE FRAME NODES

NOEUDS REPERES POUTRE no_poutre [nb_noeuds]

$$\begin{Bmatrix} \text{noeud\_depart} & \text{pas\_n} & \text{x\_loc} & \text{y\_loc} & \text{z\_loc} \\ \langle \text{no\_noeud} & \text{point\_fibre} & \text{x\_loc} & \text{y\_loc} & \text{z\_loc} \rangle_{\text{nb\_noeuds}} \end{Bmatrix}$$

Parameters

- no_poutre: number of a beam belonging to the model and obtained either by direct recording followed by eventual positioning, or by copying and positioning an existing beam;
- nb_noeuds: number of non-beam nodes described relative to its reference fiber, eventually (re)positioned in the global reference frame (via a POSITION POUTRE or COPIE POUTRE command):
  - do not provide this value if the number of nodes to be created is equal to the number of points of the reference fiber, if they are numbered by the PH1 module, in arithmetic progression, and if their local coordinates are constant; it is then a line of nodes "parallel" to the reference fiber, case 1;
  - indicate a positive value if at least one of the three conditions above is not verified, 2 cases;
- noeud_depart, pas_n: starting value (positive) and node numbering step (positive or negative), in case 1;
- no_noeud, point_fibre: node number and reference fiber point number to which it attached, in case 2; the selection order of the beam sections is not imposed;
- x_loc, y_loc, z_loc: coordinates of a node in the generic reference frame implicitly designated by its sequence number (case 1), or explicitly by point_fibre (case 2).

Whether generated or provided, the node numbers must be unique (for the full model), positive and less than 1000_000.

Functions

This command provides the numbers and local coordinates of a series of nodes described in certain beam generic reference frames.

Conditions for use

- Do not introduce this type of command if the model does not include a beam.
- If the model has at least one beam, we can introduce any number of NOEUDS REPERES POUTRE commands, which can exceed the number of beams of the model (a defined beam, with several lines of parallel nodes attached to it).
- The no_poutre number must be part of the model's reference beam list, set by the pair of commands NOEUDS FIBRE POUTRE and ELEMENTS POUTRE.

Methodological advice

- ❐ This command can be useful for simply defining the non-beam nodes of ribbed or mixed decks, modeled as several "parallel" beams, or nodes whose coordinates are easily accessible, relative to a deck roadway axis or a tower axis (stay-cable attachment points), concurrently with the EXCENTREMENTS command.

Examples

Beam number 20 comprises 5 sections; in case 1, each section produces one non-beam node, whose numbers are in arithmetic progression and local coordinates are constant.

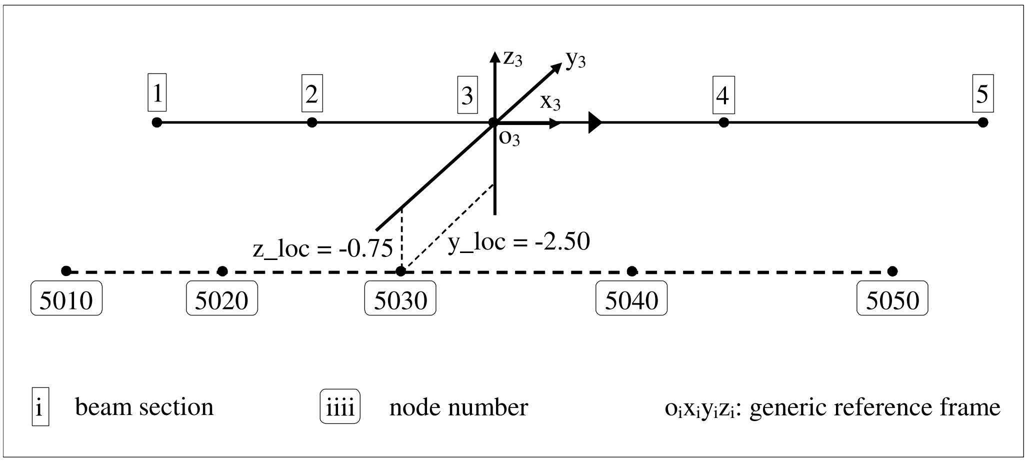


Figure 6.7 - Beam, node lines parallel to the reference fiber, case 1

```
NOEUDS REPERES POUTRE  20
$ start node  pitch_n     x_loc  y_loc  z_loc
  5010           10       0.00   -2.50  -0.75
```

In case 2, sections 1 and 5 produce a non-beam node, the third produces two, sections 2 and 4 do not produce any; their local coordinates are variable; the non-ascending selection order of the beam sections is accepted.

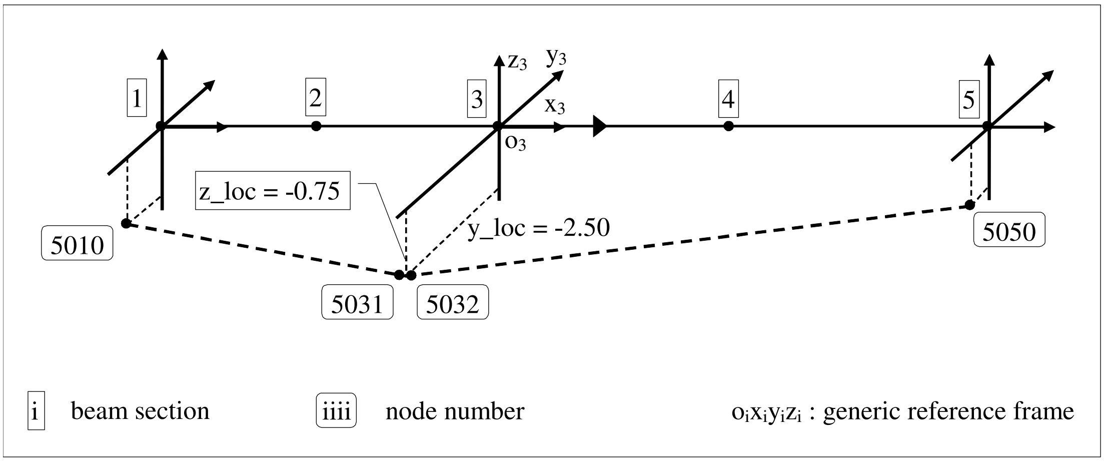


Figure 6.8 - Beam, discrete nodes in a generic reference frame, case 2

```
NOEUDS REPERES POUTRE  20  4
$ n°_node   fiber point   x_loc   y_loc   z_loc
  5010       1             0.00    -0.50   -0.75
  5050       5             0.00    -0.35   -0.75
  5031       3             0.00    -2.50   -0.75
  5032       3             0.00    -2.50   -0.75
```

Related commands

APPUI ; POSITION POUTRE ; COPIE POUTRE ; NOEUDS FIBRE POUTRE

NOEUDS REPERE GLOBAL ; ELEMENTS POUTRE ; ELEMENTS COURANTS

ELEMENTS RIGIDES ; ELEMENTS ARTICULATIONS

6.12 - GLOBAL REFERENCE FRAME NODES

NOEUDS REPERE GLOBAL nb_noeuds

$$\langle \text{no\_noeud} \quad \text{x\_noeud} \quad \text{y\_noeud} \quad \text{z\_noeud} \rangle_{\text{nb\_noeuds}}$$

Parameters

- nb_noeuds: number of non-beam nodes (other than those described using the NOEUDS REPERES POUTRE commands); this value must be positive if there is at least one beam, and greater than 1 otherwise;
- no_noeud: node number; node numbers must be unique (for the entire model), positive and less than 1000_000;
- x_noeud, y_noeud, z_noeud: coordinates of a node in the global reference frame.

Functions

This command provides the numbers and coordinates for all non-beam nodes directly described in the global reference frame.

Conditions for use

- This command is mandatory if the model does not include a beam, optional otherwise.
- Insert only one command of this type if the model has at least one node defined in the global reference frame.

Examples

```
NOEUDS REPERE GLOBAL  3
$ no_node x_node y_node z_node          no_node x_node y_node z_node
  2000     0.00    0.00    1.00         2025     0.00    0.00    2.00
  2050     0.00    0.00    3.00
```

Related commands

APPUI ; NOEUDS FIBRE POUTRE ; NOEUDS REPERES POUTRE

ELEMENTS COURANTS ; ELEMENTS RIGIDES ; ELEMENTS ARTICULATIONS

6.13 - BEAM ELEMENTS

ELEMENTS POUTRE no_poutre

$$\left\{ \begin{array}{l} \text{element\_depart} \quad [\text{pas\_e}] \\ \langle \text{no\_element} \quad \text{noeud\_debut} \quad \text{noeud\_fin} \rangle_{\text{nb\_pointsf\_}-1} \end{array} \right\}$$

Parameters

- no_poutre: number of a beam belonging to the model and obtained either by direct recording followed by eventual positioning, or by copying and repositioning an existing beam.

The number of geometric elements created by the introduction of a beam is always equal to the number of its reference fiber points minus one $(\text{nb\_points\_f}-1)$.

If we are in case 1 of the NOEUDS FIBRE POUTRE no_poutre command, the elements are assumed to be numbered in arithmetic progression and their numbers and incidence are determined by the PH1 module; in case 2, element numbers and incidence are provided.

- element_depart, pas_e: starting value (positive) and element numbering step (positive or negative, to be provided only if the number of beam sections exceeds 2), in case 1;
- no_element: beam element number, in case 2, positive;
- noeud_debut, noeud_fin: start and end node numbers (which must exist, and for the correct sections, in the list of the NOEUDS FIBRE POUTRE no_poutre command), in case 2.

In both cases, all the beam elements are listed in ascending order of section numbers (direction of travel). Whether generated or provided, the numbers of elements must be unique (for the full model) and less than 1000_000.

Functions

This command provides the numbers and incidence of the elements of a beam in the model.

Conditions of use

- Do not introduce this type of command if the model does not include a beam.
- If the model has at least one beam, provide a command of this type for each beam belonging to it (present directly or obtained by copying).
- NOEUDS FIBRE POUTRE and ELEMENTS POUTRE command groups (in equal numbers) determine the number of beams belonging to the model, and the reference list of their numbers (identical, in any order, from one group to another).

Methodological advice

- If the nodes of the mean fiber of a beam conform to case 1, and if its elements are not numbered regularly, provide the number and the full list of node numbers in the associated NOEUDS FIBRE POUTRE command (forced into case 2).

Examples

Beam number 20 comprises 5 sections and 4 elements; in case 1, each section produces only one node (very common); their numbers are in arithmetic progression.; the elements are numbered in the same way, their incidence is determined by the PH1 module.

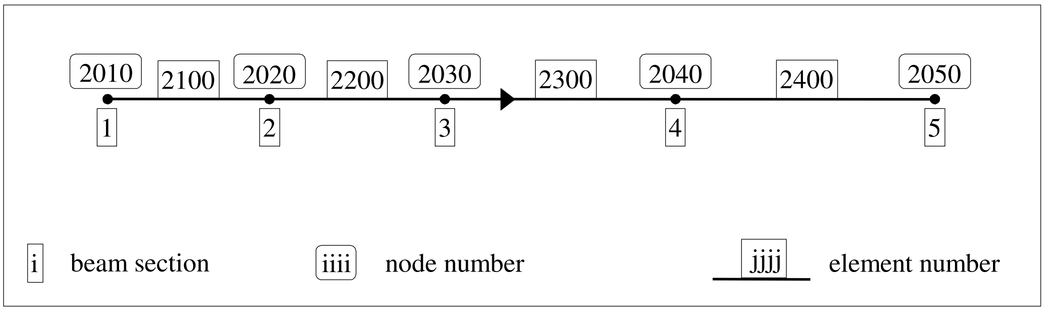


Figure 6.9 - Beam, mean fiber nodes and elements, case 1

```
NOEUDS FIBRE POUTRE  20
$ start node  pitch_n
  2010          10
ELEMENTS POUTRE  20
$ start element  pitch_e
  2100             100
```

In case 2, each section produces a node, except for the third which is duplicated; all node numbers are provided; element numbers, although regularly spaced, are also provided, as well as their incidence.

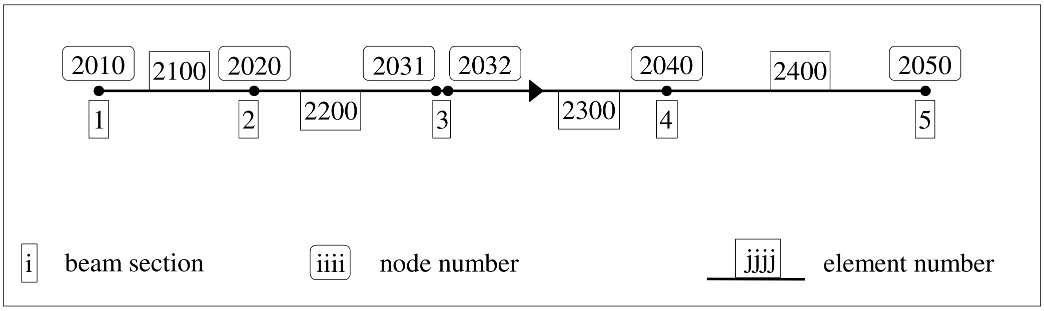


Figure 6.10 - Beam, mean fiber nodes and elements, case 2

```
NOEUDS FIBRE POUTRE  20  6
$ n°_nodes_s  no_node  no_node     n°_nodes_s  no_node  no_node
  1            2010                  1           2020
  2            2031      2032        1           2040
  1            2050
ELEMENTS POUTRE  20
$ n°_element  start node  end node     n°_element  start node  end node
  2100        2010         2020           2200        2020         2031
  2300        2032         2040           2400        2040         2050
```

Related commands

EQUIPAGE MOBILE ; POSITION POUTRE ; COPIE POUTRE

NOEUDS FIBRE POUTRE ; NOEUDS REPERES POUTRE ; EXCENTREMENTS

6.14 – CURRENT ELEMENTS

$$\text{ELEMENTS}\quad\text{COURANTS}\quad\begin{Bmatrix}\text{GROUPES}\\ \text{INDIVIDUALISES}\end{Bmatrix}\quad\text{nb\_elements}$$

$$\left\{\begin{matrix}\left\langle \text{no\_element}\quad\text{noeud\_debut}\quad\text{noeud\_fin}\right\rangle_{\text{nb\_elements}}\quad\beta\quad\text{nom\_materiau}\quad\text{nom\_section}\\ \left\langle \text{no\_element}\quad\text{noeud\_debut}\quad\text{noeud\_fin}\quad\beta\quad\text{nom\_materiau}\quad\text{nom\_section}\right\rangle_{\text{nb\_elements}}\end{matrix}\right\}$$

Parameters

- nb_elements: number of current elements of the model, positive;
- no_element: current element number, positive; the numbers of elements must be unique (for the full model) and less than 1000_000;
- noeud_debut, noeud_fin: start and end node numbers, described on a NOEUDS FIBRE POUTRE, NOEUDS REPERES POUTRE or NOEUDS REPERE GLOBAL command;
- β : Angle of rotation around the Gx axis of an element, positioning the Gyz principal reference frame of its cross section, relative to the global reference frame; the original position of the element (corresponding to a zero value of β) depends on the vertical absolute angle of inclination, α_v, of its Gx axis (parallel to the OZ axis of the global reference frame, Figure 6.11); the element is assumed to be vertical when α_v is less than 0.0001 rd; there are two possible cases:
  - adding 1000.0 to the absolute value of β (keeping its sign) is the same as designating a vertical element; in this case, an error message will appear when α_v is greater than or equal to 0.01 rd;
  - providing a "true" value of β is the same as designating a non-vertical element; in this case, a warning message will appear when α_v is less than 0.01 rd.

In both cases, when α_v is greater than or equal to 0.0001 rd and less than 0.01 rd, the angle β is corrected by subtracting the calculated angle between the OX axis and the projection of the Gx axis on the plane OXY.

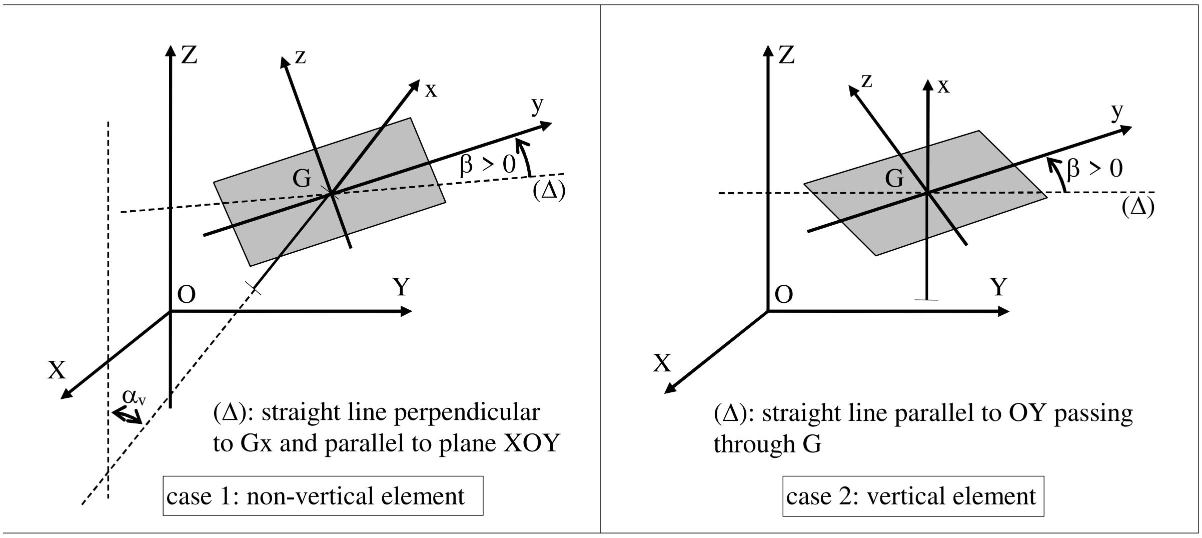


Figure 6.11 - Current element, angle β of rotation around Gx

- nom_materiau: name of the assigned material, defined on a MATERIAU command (for non-beam elements);
- nom_section: name of the assigned section-type, defined on a SECTION TYPE command.

If the angles β, the materials and the section-types assigned are the same, they can be factorized (GROUPES option), otherwise (INDIVIDUALISES option), they must be provided individually.

Functions

This command provides the numbers, incidence, rotation angles, materials, and section-types assigned to a group of current elements (non-beam elements that are neither rigid nor articulations).

Conditions for use

- This command is mandatory if the model does not include a beam, optional otherwise.
- If the model has at least one current element, provide at least one command of this type.
- The length of a current element must be greater than 0.01.

Methodological advice

- See SECTION TYPE command.

Examples

```
$ elements of a prismatic pile with medium rectilinear and vertical fiber
ELEMENTS COURANTS GROUPES  3
$ n°_element start node end node
  10801      1405        1410
  10802      1410        1415
  10803      1415        1420
-1010.0  BETON_T  S_PILE      $ vertical elements, beta = -10.0

$ elements of a variable-section pile
$ with medium rectilinear and vertical fiber
ELEMENTS COURANTS INDIVIDUALISES  3
$ n°_element start node   end node    beta     material name   section name
    10801       1405       1410    -1010.00       BETON_T        PILE_1
    10802       1410       1415    -1010.00       BETON_T        PILE_2
    10803       1415       1420    -1010.00       BETON_T        PILE_3

$ group of non-vertical elements with different characteristics
ELEMENTS COURANTS INDIVIDUALISES  3
$ n°_element  start node  end node    beta     material name   section name
     1001        105        110       90.00       BETON_T        LIAISON
     1002        110        115        0.00       ACIER_HR       HAUBAN
     1003        115        120       90.00       BETON_T        LIAISON
```

Related commands

MATERIAU ; SECTION TYPE ; NOEUDS FIBRE POUTRE

NOEUDS REPERES POUTRE ; NOEUDS REPERE GLOBAL ; EXCENTREMENTS

6.15 - RIGID ELEMENTS

ELEMENTS RIGIDES nb_elements

$\langle \text{no\_element} \quad \text{noeud\_debut} \quad \text{noeud\_fin} \rangle_{\text{nb\_elements}}$

Parameters

- nb_elements: number of rigid elements of the model, positive;
- no_element: rigid element number:
  - a positive value means a "visible" element processed by mechanical modules, such as beams or current elements; the numbers of positive elements must be unique (for the full model) and less than 1000_000;
  - a zero value means an "invisible" element, that can be implicitly activated by contact with an activated element or a positioned articulation-type; results concerning it are not editable;
- noeud_debut, noeud_fin: start and end node numbers, described on a NOEUDS FIBRE POUTRE, NOEUDS REPERES POUTRE or NOEUDS REPERE GLOBAL command.

Functions

This command provides the numbers and incidence of all explicitly defined visible or invisible infinitely rigid elements without assignment of section-type or material.

Their mechanical characteristics are determined by the PH3 module, based on amplification of the maximum characteristics of the beams or current elements.

Conditions for use

- ☐ If the model has at least one explicitly defined rigid element, provide a single command of this type.
- ☐ The length of a rigid element must be greater than 0.01.
- ☐ It is not permitted to apply loads on visible or invisible rigid elements.

Methodological advice

- ☐ See SECTION TYPE command.

Examples

```
ELEMENTS RIGIDES  4
$ n°_element  start node end node    n°_element   start node   end node
  20801       12405        12410        20802       12410        12415
  0           12415        12420        0           12420        12425
```

Related commands

NOEUDS FIBRE POUTRE ; NOEUDS REPERES POUTRE

NOEUDS REPERE GLOBAL

6.16 - ARTICULATION ELEMENTS

ELEMENTS ARTICULATIONS nb_elements

$\langle \text{no\_element} \quad \text{noeud\_debut} \quad \text{noeud\_fin} \rangle_{\text{nb\_elements}}$

Parameters

- nb_elements: number of articulation elements of the model, positive;
- no_element: articulation element number, positive; the numbers of elements must be unique (for the full model) and less than 1000_000;
- noeud_debut, noeud_fin: start and end node numbers, described on a NOEUDS FIBRE POUTRE, NOEUDS REPERES POUTRE or NOEUDS REPERE GLOBAL command.

Functions

This command provides the numbers and incidence of all articulation elements (ad hoc) without articulation-type assignment.

Conditions for use

- ❐ If the model has at least one articulation element, provide a single command of this type.
- ❐ The length of an articulation element must be less than 0.01.

Methodological advice

- ❐ The presence of articulation elements may be imposed by the use of specific commands of the PH3 module, or certain construction modes (incremental launching, etc.).

Examples

```
ELEMENTS ARTICULATIONS  2
$ n°_element  start node  end node
  30501       52405        52410
  30502       52415        52420
```

Related commands

ARTICULATION ; NOEUDS FIBRE POUTRE ; NOEUDS REPERES POUTRE

NOEUDS REPERE GLOBAL

6.17 - ECCENTRICITY

$$\text{EXCENTREMENTS}\left\{\begin{array}{l}\text{ORIGINE}\\ \text{EXTREMITE}\end{array}\right\}\left\{\begin{array}{l}\text{CONSTANTS}\\ \text{VARIABLES}\end{array}\right\}\left\{\begin{array}{l}\text{GLOBAL}\\ \text{POUTRE}\quad\text{no\_poutre}\end{array}\right\}\text{nb\_exe}$$

$$\left\{\begin{array}{l}\left\langle\text{no\_element}\quad[\text{point\_fib re}]\right\rangle_{\text{nb\_exe}}\quad\text{exc\_x}\quad\text{exc\_y}\quad\text{exc\_z}\\ \left\langle\text{no\_element}\quad[\text{point\_fib re}]\quad\text{exc\_x}\quad\text{exc\_y}\quad\text{exc\_z}\right\rangle_{\text{nb\_exe}}\end{array}\right\}$$

Parameters

- no_poutre: number of a beam belonging to the model, for which the reference fiber generic reference frames, eventually replaced in the global reference frame (by positioning or copying), serve as a reference to express the eccentricity, if working in a beam generic reference frame (POUTRE option);
- nb_exe: number of current elements to be offset, positive;
- no_element: number of an element, belonging to the ELEMENTS COURANTS command numbers list;
- point_fibre: no_poutre beam reference fiber point number, in the generic reference frame in which the eccentricities are provided, when using the POUTRE option;
- exc_x, exc_y, exc_z: eccentricity valuesof an element, at its origin (ORIGINE option) or at its end (EXTREMITE option), expressed in a translated global reference frame, at the origin or end node of the element (GLOBAL option); new coordinates of its origin or end node in the point_fibre generic reference frame of the no_poutre beam, when using the POUTRE option.

If the eccentricity values are constant (CONSTANTS option), they can be factorized, otherwise (VARIABLE option), three valuesmust be provided per element to be offset.

Functions

This command is to offset a group of current elements, at their origin or end, by introducing unnumbered nodes (that are indirectly loadable and accessible for displacement) and implicitly defined invisible rigid elements.

Eccentricities can be expressed in the global reference frame, or in certain beam generic reference frames.

Conditions for use

- Do not introduce a command of this type if the model has no current elements.
- If the model has at least one current element, insert a command of this type per group of elements to be offset.
- It is not possible to offset rigid beam elements or articulations.
- It is not possible to offset an element more than once, at the origin or at the end.
- If certain eccentricities are provided in beam reference frames, the model must provide for this and the no_poutre number must be included in the model's reference beam list, set by the pair of commands NOEUDS FIBRE POUTRE and ELEMENTS POUTRE.

Examples

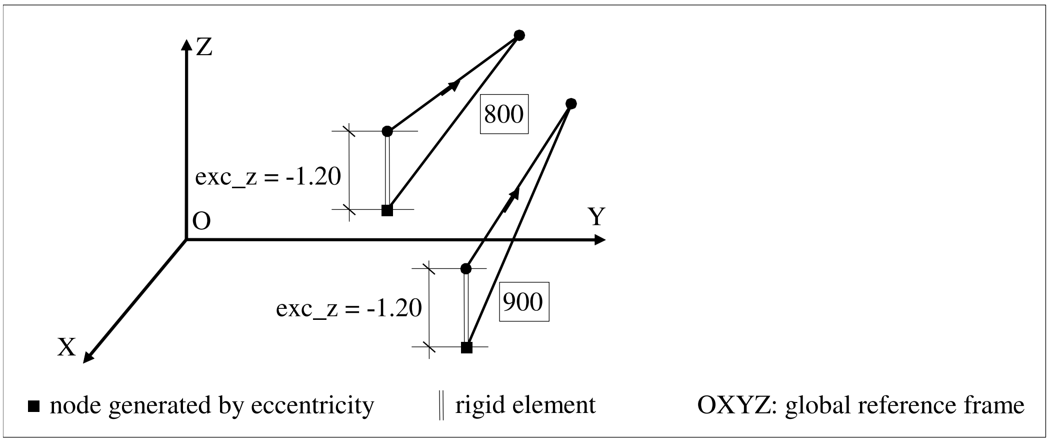


Figure 6.12 - Constant eccentricities at the origin, in the global reference frame

```
EXCENTREMENTS ORIGINE CONSTANTS GLOBAL  2
$ n°_element  n°_element
  700         800
$ exc_x  exc_y  exc_z
  0.00   0.00   -1.20
```

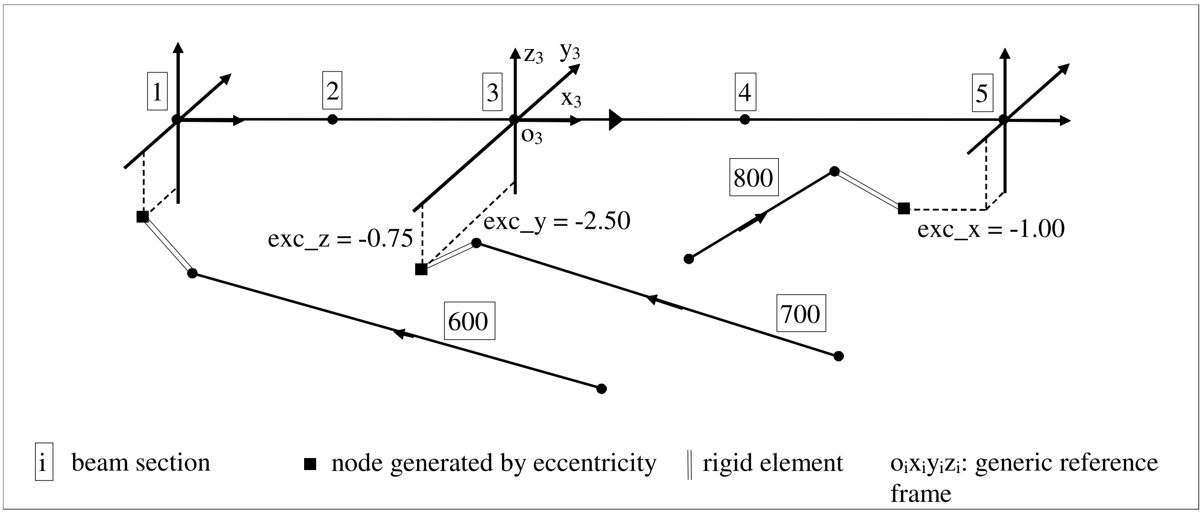


Figure 6.13 - Variable eccentricities at the end, in a beam generic reference frame

```
EXCENTREMENTS EXTREMITE VARIABLES POUTRE  20  3
$ n°_element  fiber point  exc_x  exc_y  exc_z
  600         1             0.00  -0.50  -0.75
  800         5            -1.00  -0.35  -0.75
  700         3             0.00  -2.50  -0.75
```

Related commands

NOEUDS FIBRE POUTRE ; ELEMENTS POUTRE ; ELEMENTS COURANTS

6.18 - END

FIN

Functions

This command marks the end of the model data.

Conditions of use

- Must be the last line of the command file.

Examples

```
MODELE
.......
.......
FIN
```

Related commands

MODELE

Chapter 7

Drawing

TRAI RAP

LABL LEG / LABL RES

IMPR TOT / IMPR 4X4 / IMPR ZON / IMPR ECH / IMPR NOR

DESS OPT

7.2 - ACTIONS

DECOUPER

CHOISIR

RECOLLER

ZOOMER

RESITUER

FIXE ECH

DEPLACER

CHERCHER

TOURNER

RECADRER

AMPL DEF

Recommendations for use

ECHE RES

LONG LAB

POINTER

GOMMER V

XY>XZ>YZ

PROJETER

CHGT IMP

IMPRIMER

Introduction

The PH2 module is responsible for representing the proposed model, along with certain related results of calculations, in graphical format. This chapter describes the features and design options, how to implement them, and their likely effects.

The standard graphical representation provided may include:

- an image with control of attributes of the STRUCTURE INITIALE (undeformed) frozen in a static diagram saved by the PH3 module during the simulation of the construction process (with certain support-types and articulation--types placed); active and inactive elements are managed separately;
- an image with control of attributes of the active STRUCTURE DEFORMEE (applying to a real state of deformation or to a buckling or vibration eigenmode), with amplified node displacements (their rotations are not taken into account);
- a group of COURBES from calculation results displayed for certain ends of selected active elements (forces in element or section reference frames, normal and/or tangential stresses).

The drawing can be reduced to a SOUS-STRUCTURE obtained using plans for cutting and/or selection of groups of elements.

All attribute details are displayed in options; their visibility is linked to that of the structural parts to which they relate; their sizes, fixed in actual values for each type of graphic material, can be enlarged by magnifying effect.

Four view-types, whose projection planes can be modified, are stored: perspective, top view, elevation and view from the right. They can be run in sequence if the selected display mode is SIMPLE, or displayed simultaneously and swapped, if QUADRUPLE (only one type of view will be ACTIVE with control of attributes).

The width and height scale factors (calculated or imposed) can be equal or different (ECHELLES EGALES or ECHELLES DIFFERENTES mode).

Auto reframing functions take into account the initial structure, the deformed structure and the scale factor chosen to transform the lengths of the valuesof results at element ends.

These functions are activated when framing must offer the best conditions of visibility for all that is demanded, it then works in FENETRE TOTALE mode. They are deactivated when the full window is no longer in force (focus on one part of the drawing, for example), it then works in FENETRE PARTIELLE mode.

Operating principles

To represent the model with deformed structure or curves, you must run the RES module and choose STRUCTURE in its main menu (competing with the GRAPHE option), after selecting the desired ETUDES and EFFETS (full version).

The complete model geometry drawing can be obtained independently, under the following conditions (reduced version):

- all its elements are considered active;
- no support-type and no articulation--type are positioned;
- the representative functions of the deformed structure and results at the element ends are disabled.

Both versions work identically, interactively, so with no need to provide a data file. Additional data is keyed in and choices and plotting are performed using the mouse.

For the RES module, at the first pass through the STRUCTURE choice, the complete model is presented, on a single perspective view, with a deformed structure or group of curves. The attribute details are reduced to certain presentation legends. On subsequent passes, the continued presence of the display, scale factor calculation, attributes and printing options is ensured. For the reduced version, the complete model is presented, by default, analogously.

Consistency and appearance of drawings

The drawings to be produced on the ECRAN or to be printed on the TRACEUR may include:

- an image of the projected global reference frame (direct reference frame, always present);
- squares marking the non-support nodes of the initial and deformed structures, and triangles marking the support nodes of the initial structure;
- node numbers of the initial or deformed structure; displacement valuescan be displayed for certain nodes of the deformed structure;
- the legend of non-zero plotted displacements, with amplifying coefficient;
- images of projected support reference frames and names of the placed support--types, displayed on the initial structure;
- straight mean lines of the initial and deformed elements; the color denotes the class; the type of line distinguishes the initial active elements, initial inactive elements and deformed elements (still active);
- the structure legend attesting to the use of line attributes for the elements present in the display;
- numbers, section-type names or material-type names assigned to the initial or deformed elements;
- direction arrows for the initial elements;
- projected yz reference frames of the initial elements;
- curves representing the results shown at the ends of certain active initial elements, with extension lines; the pairs of envelope curves have identical representation attributes; the type and thickness of the line replacing the color on monochrome plotters; certain valuesmay be plotted;
- identification labels shown on curves or in a results legend.

Complete list of functions

Functions marked "*" are disabled in the reduced version.

- display or delete the search viewfinder on the screen;
- passage from single display mode on the screen, to quad mode, or vice versa;
- passage from same scales mode to different scales mode, or vice versa;
- display or hide initial active or inactive elements;
- display or remove the deformed structure *;
- emphasize the deformed structure with respect to the initial one, and vice versa *;
- optional display of a structure legend;
- optional display of node markers;
- optional display of node numbers;
- optional display of support reference frames and support--type names;
- optional display of numbers, section-type names or material-type names for initial or deformed elements;
- optional direction marking of initial elements;
- optional display of yz reference frames on initial elements;
- optional display of results at the ends of certain active initial elements *;
- transfer of curve identification labels to the results legend, or vice versa *;
- choice of a plotter print mode;
- lock or unlock the immediate application of display option changes;
- current substructure cut by a plane and temporary removal of nodes and elements of a half-space;
- choice of elements defining the current substructure, to be added or removed;
- passing from a substructure to the complete structure;
- enlarging a rectangular part of the drawing (zoom), or the size of the attribute details, with the option to return to normal size (magnifying glass);
- moving the drawing nearer or farther away, with preservation of its central point;
- imposition of scale factors;
- moving the current window with conservation of scale factors;
- search for a node or element which is then placed in the center of the drawing;
- rotation of the drawing in its plane;
- reframing the drawing (return to full window);
- modification of the displacement amplifying coefficient for a deformed structure *;
- changing the results scale factor *;
- changing the maximum effective length of curve identification labels *;
- plotting and displaying values on curves, or node displacements, with specific markers; erasing a plotted value *;
- erasing plotted values, on curves, or node displacements *;
- rapid transition from the active view-type to one of the other three view-types;
- changing the orientation angles of the active view-type projection plane;
- changing the name of the storage file for the outputs to be printed;
- print all or part of the active view-type, or the four view-types.

Standard screen

The screen is subdivided into the following five areas:

- title area;
- model drawing area;
- error message area, informative, or data input request;
- subtitle and data input area;
- menu area consisting of a group of OPTIONS boxes and a group of ACTIONS boxes (there are no sub-menus).

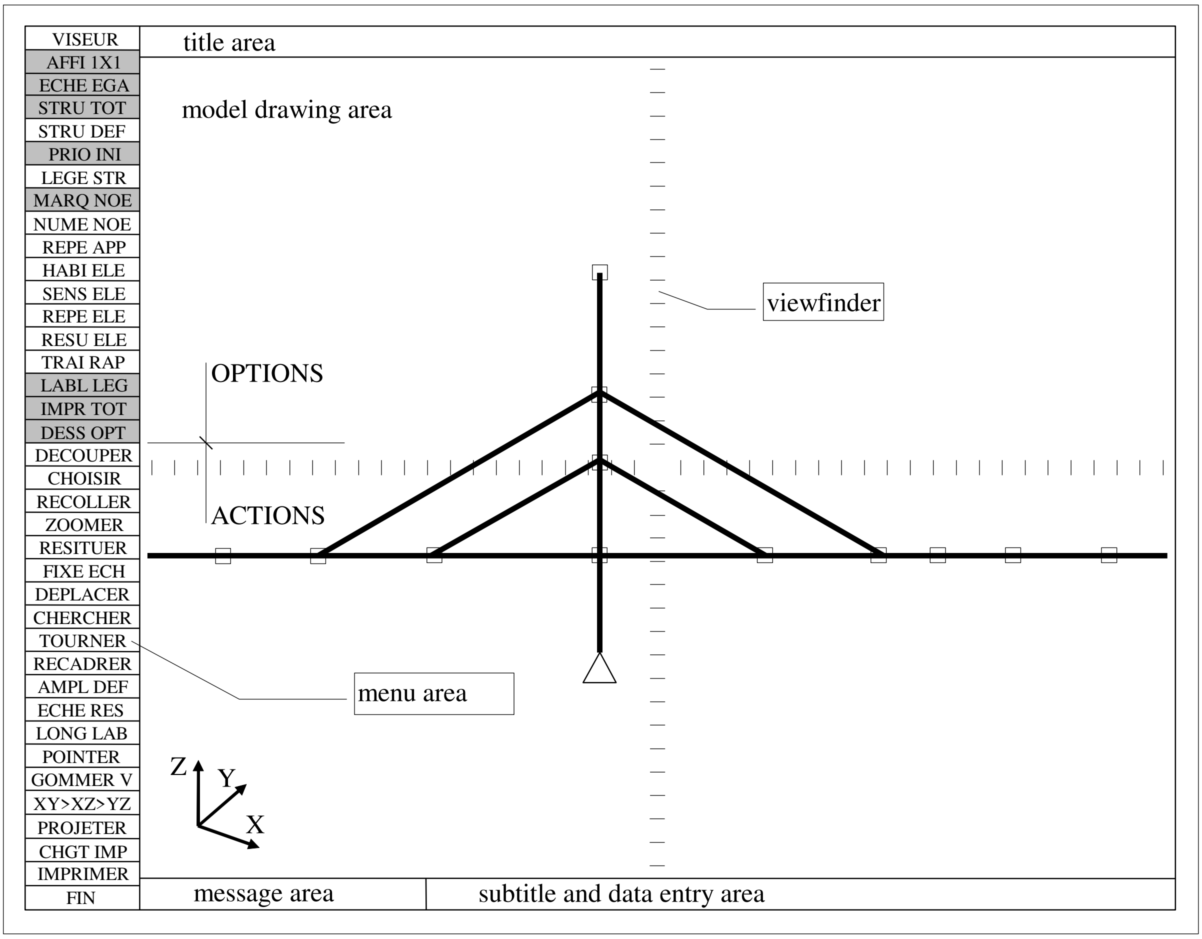


Figure 7.1 - Standard screen subdivisions

Title area

Contains, on a line, the following headings:

- the short title of the organization operating the software (five characters);
- --PCP-PH2;
- current date and time;
- main title of the model (compressed);
- the orientation angles of the current projection plane;
- the calculated or imposed scale factors, in width and height, preceded by '1/' and symbolized in some subsequent diagrams by ai, bi, ci and di.

Model drawing area

With the title area, it can accommodate one or four view-types; for quad display, the title area is repeated.

Each view-type is equipped with a safety edge whose width depends on the windowing mode setting and the actual size of certain attribute details.

The width/height ratio of this area determines the corresponding scale factors, when working in different scales mode and full window.

Message area

The error or information messages will appear on a red background, those requesting data entry will appear on a green background.

Subtitle and data entry area

It is necessary to enter the data designated in the message area.

The keywords are converted to uppercase, if applicable, and recognized by their first four letters. Numerical valuesare checked (number of fields, readability, veracity of their contents in relation to certain tolerances).

If an error message appears, this area is reset and you must enter new valuesor press the ESC key before initiating a new action.

Menu, OPTIONS boxes

The background color is green if an option is inactive and red if it is active.

"Multiple" options such as screen sharing, scale factor calculation modes, priority allocation to the deformed structure, placement of identification labels of curves and choice of print mode remain active.

The changeover from one state to another for a single option, or from one choice to another for multiple options, is made by simply clicking in the appropriate box.

Menu - ACTION boxes

Move the pointer and simply click on the appropriate box or press the ENTER key, to initiate an action.

The ESC key allows the interruption of an ongoing action.

Since the pointer comes back to the box selected after performing an action, it is possible to repeat it by pressing the ENTER key without reusing the mouse.

Menu texts

Here is the complete list of texts that may appear on the menu, in their order of appearance, and their meanings.

Text	meaning
VISEUR	Management of search viewfinder display
AFFI 1X1	The model is shown in a single view-type
AFFI 4X4	The model is shown in four simultaneous view-types
ECHE EGA	The scale factors are equal in width and height
ECHE DIF	The scale factors may be different in width and height
STRU INI	The initial structure is not drawn
STRU ACT	Only the active elements of the initial structure are drawn
STRU TOT	The full initial structure (active and inactive elements) is drawn
STRU DEF	Management of the display of the deformed structure
PRIO INI	If it appears, the initial structure, now a priority, is strengthened and with control of attributes
PRIO DEF	If it appears, the deformed structure, now a priority, is strengthened with control of attributes
LEGE STR	Management of the display of the structure legend
MARQ NOE	Management of the display of node markers
NUME NOE	Management of the display of node numbers
REPE APP	Management of the display of support reference frames and support-type names
HABI ELE	Element attributes are disabled
NUME ELE	The element numbers are displayed on the initial/deformed structure
SECT ELE	Element section-type names are displayed on the initial/deformed structure
MATE ELE	Element material-type names are displayed on the initial/deformed structure
SENS ELE	Management of the direction marking of initial elements
REPE ELE	Management of the display of yz reference frames on initial elements
RESU ELE	Management of the display of results at element ends
TRAI RAP	Management of the display of extension lines on curves
LABL LEG	The curve identification labels are shown in the results legend
LABL RES	The curve identification labels are dispersed on the curves
IMPR TOT	Single full printing, calculated scale factors, saturated plotter
IMPR 4X4	Quad full printing, calculated scale factors, saturated plotter
IMPR ZON	Single partial printing, calculated scale factors, saturated plotter
IMPR ECH	Single partial printing, imposed scale factors, saturated plotter
IMPR NOR	Standardized printing with title block, imposed scale factors, unsaturated plotter
DESS OPT	Management of the immediate application of display option changes

Table 7.1 - Menu, options

Only the ECHE EGA, ECHE DIF, STRU INI, STRU ACT, STRU TOT, STRU DEF, PRIO INI and PRIO DEF options affect the four view-types in quad display mode.

Text	**Meaning**
DECOUPER	Current substructure cut by a plane
CHOISIR	Select elements to define or change the current substructure
RECOLLER	Pass from a substructure to the complete structure
ZOOMER	Enlarge a part of a drawing * or attribute details
RESITUER	Move the drawing nearer or farther away *
FIXE ECH	Imposition of scale factors *
DEPLACER	Move the current window *
CHERCHER	Search for a node or element *
TOURNER	Rotate the drawing in its plane
RECADRER	Reframe the drawing **
AMPL DEF	Change the node displacement amplifying coefficient
ECHE RES	Change the results scale factor
LONG LAB	Change the maximum effective length of curve identification labels
POINTER	Plot curve values,or node displacements
GOMMER V	Clear plotted values on curves, or node displacements
XY>XZ>YZ	Quick progression of the four view-types **
PROJETER	Change the projection plane of the active view-type **
CHGT IMP	Rename the print file
IMPRIMER	Print all or part of the drawing
FIN	End of processing (reduced version), continuation of processing (full version)

(*) passage en mode fenêtre partielle ; (**) passage en mode fenêtre totale

Table 7.2 - Menu, actions

Only the DECOUPER, CHOISIR, RECOLLER and XY>XZ>YZ actions affect the four view-types in quad display mode.

7.1 - OPTIONS

VISEUR

This option manages the display of the graduated viewfinder marking the center of the model drawing area, on screen only (Figure 7.1).

For quad display (see AFFI 4X4 option), only the active view-type receives the viewfinder (recognition mark).

The CHERCHER action uses the viewfinder center as an endpoint.

AFFI 1X1 / AFFI 4X4

This option causes the display to pass from single (1X1) to quad mode (4x4), sharing the model drawing area (associated with the title area), in four quadrants of the same size, or vice versa.

When passing from 1X1 to 4X4 mode, the initial single view-type is placed in the lower left corner, and is only active for entries and attributes. Its window is not altered and any rotations (see TOURNER action) are retained, if their number does not exceed 10. The scale factors are recalculated.

The other three view-types (full, passive and no attributes, but with their own titles) surround the active view-type in the order they were generated by the XY>XZ>YZ action, turning in the counterclockwise direction.

The method of calculating the scale factors in force for the initial single view-type is applied to the four view-types (see ECHE EGA / ECHE DIF option).

Changes of options or actions involved in 4X4 mode may cause an update of the active view-type alone, or of the four view-types.

The print mode is not related to the display mode, you can obtain quad printing (see IMPR 4X4 option) in single display mode on the screen, or vice versa.

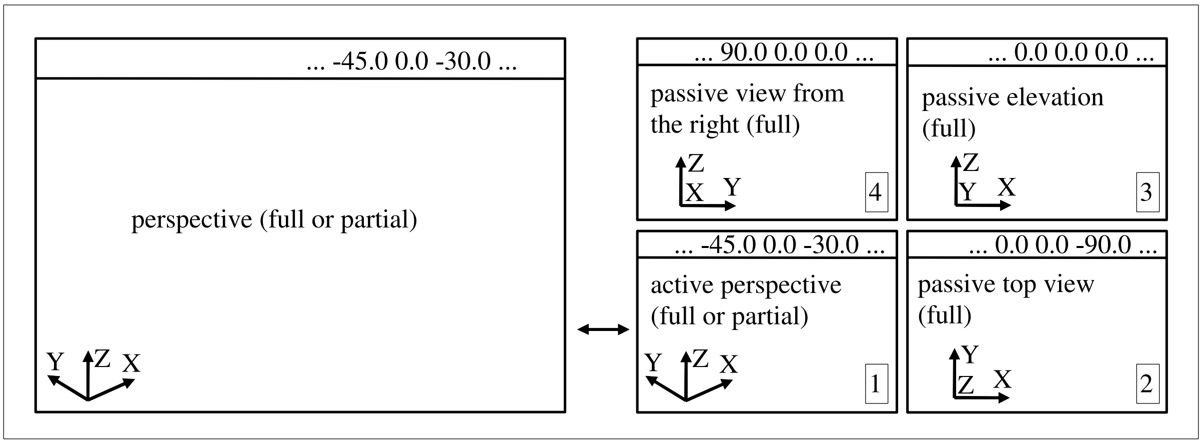


Figure 7.2 - Four view-types from a perspective

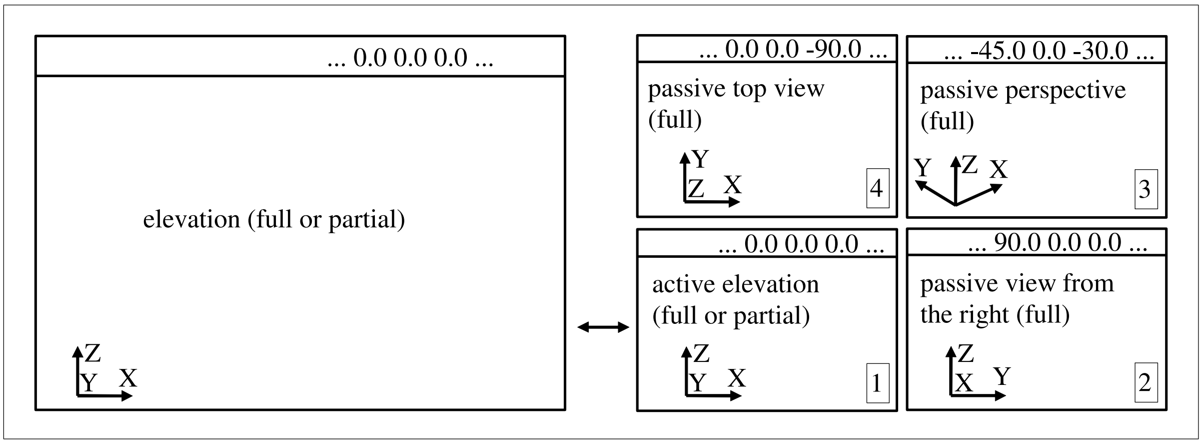


Figure 7.3 - Four view-types from an elevation

ECHE EGA / ECHE DIF

This option concerns outputs and entries on screen, and influences the plotter outputs for some print modes (see IMPR TOT / IMPR 4X4 / IMPR ZON option).

The mode of consideration of scale factors (calculated or imposed) influences the appearance of the initial and deformed structures, and curves representative of the results at the ends of elements.

By default, the drawings are produced with scale factors equal in width and height, the model drawing area being filled, generally, in only one direction; proportions and angles are preserved (ECHE EGA mode).

If ECHE DIF mode is enabled, the drawings are produced with scale factors that may be different in width and height, the model drawing area is filled in both directions; proportions and angles are not preserved.

In full window mode, auto reframing occurs on the nodes, elements and results displayed.

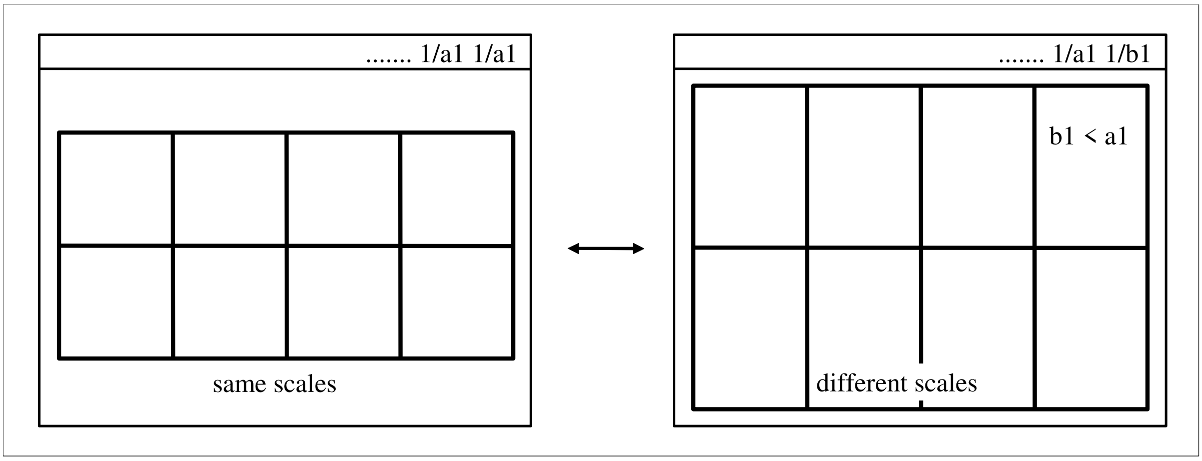


Figure 7.4 - Same and different scales, full window (predominant width)

In partial window mode, the drawing is expanded in width or height.

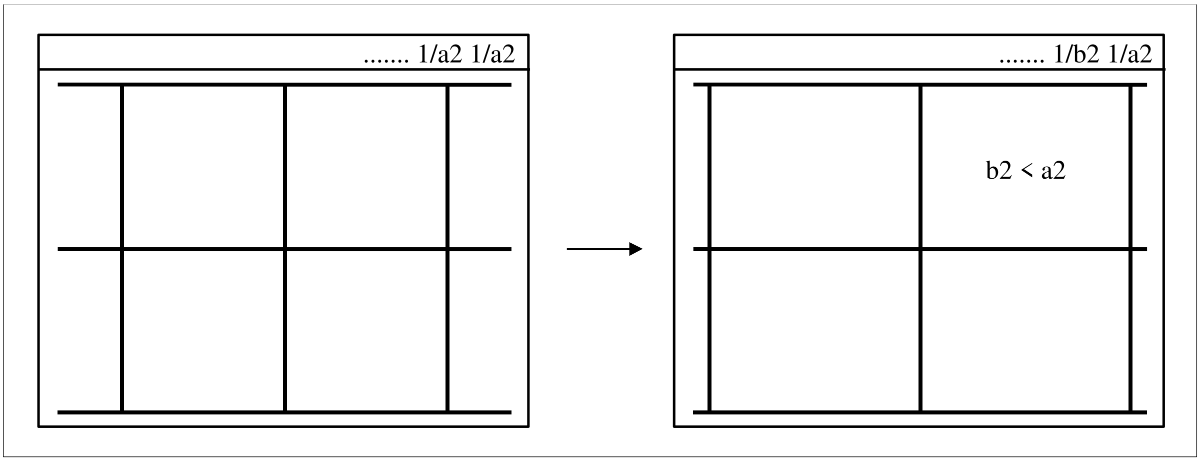


Figure 7.5 - Same and different scales, partial window, possible behavior

STRU INI / STRU ACT / STRU TOT

This option controls the level of representation of nodes and elements of the initial structure (undeformed).

The active initial structure includes non-articulation elements activated by the PH3 module (explicitly or implicitly), and the articulation elements with the remaining placed articulations-types, and the active nodes.

The following are declared active: end nodes of active elements, and the isolated nodes with the remaining placed support-types.

There are three options for displaying the initial structure (drawing only inactive elements is not considered useful), which cyclically occur in this order:

- the initial structure is not drawn (STRU INI);
- only the active initial nodes and elements are drawn (STRU ACT);
- active and inactive initial elements and nodes are drawn (STRU TOT).

Only the nodes and elements of the current substructure are represented.

In full window mode, auto reframing occurs on the nodes, elements and results displayed.

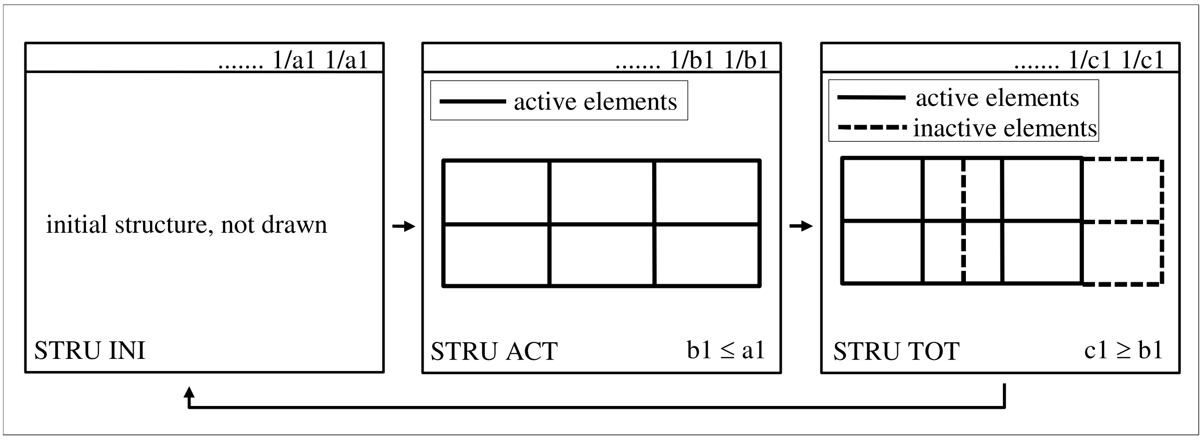


Figure 7.6 - Three states of the initial structure in full window (same scales)

In partial window mode, framing and scale factors are not altered.

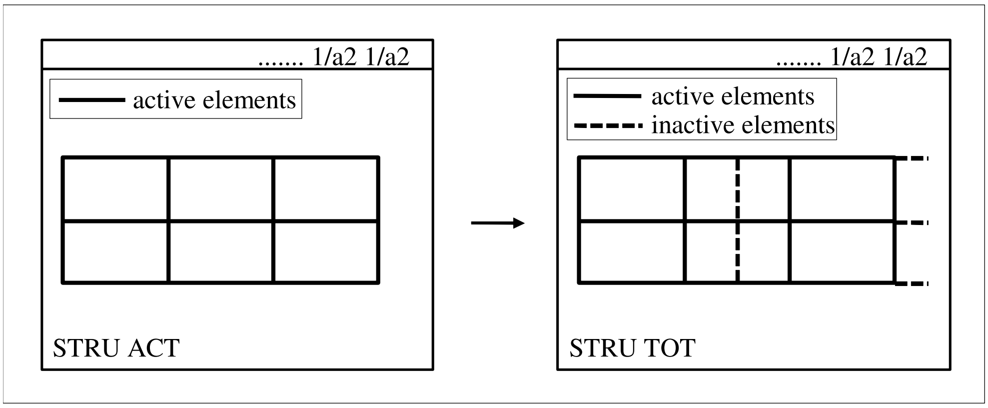


Figure 7.7 - Passing from STRU ACT to STRU TOT in partial window (same scales)

STRU DEF

This option manages the display of nodes and elements of the deformed structure; it is disabled in the absence of a deformed structure.

The deformed structure includes displaced active elements; a displacement amplifying coefficient is applied to their end nodes (can be modified using the AMPL DEF action).

It can be drawn with or without the initial structure, but never with the results at the ends of elements.

Only the nodes and elements of the current substructure are represented.

In full window mode, auto reframing occurs on the nodes and elements displayed.

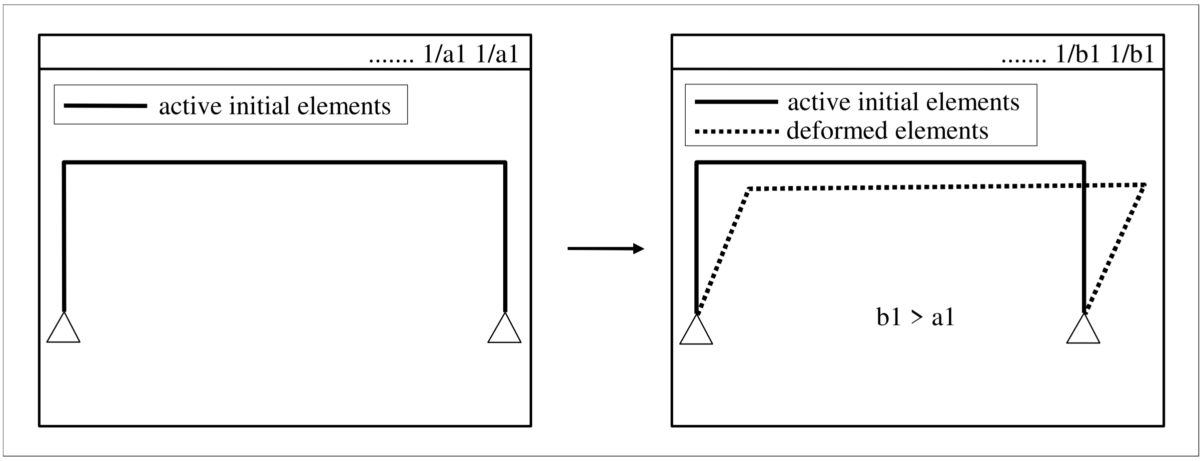


Figure 7.8 - Addition of the deformed structure in full window (same scales)

In partial window mode, framing and scale factors are not altered.

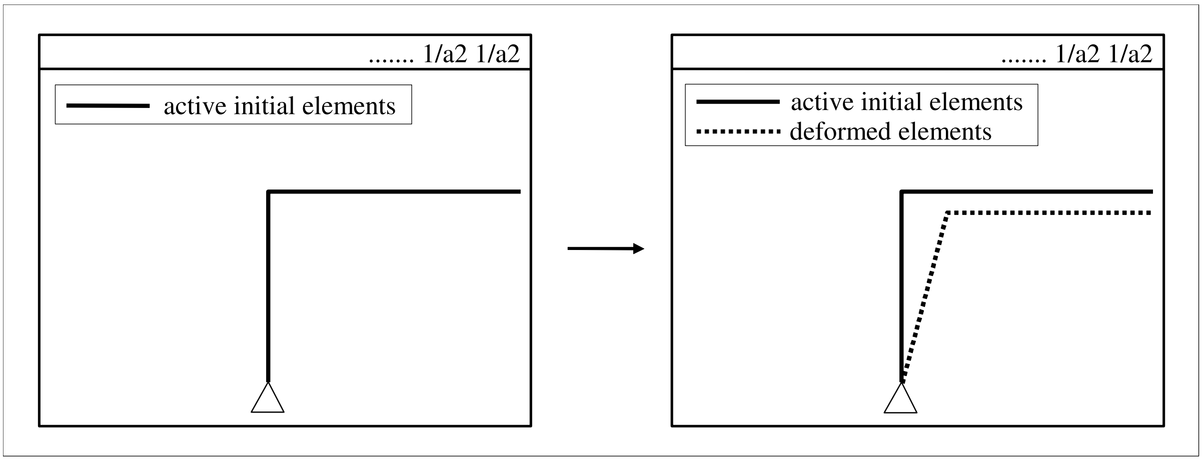


Figure 7.9 - Addition of the deformed structure in partial window (same scales)

PRIO INI / PRIO DEF

By default, the initial and deformed structures use these representation conventions:

- active initial elements are drawn in solid lines and deformed elements are drawn in dotted lines;
- when the initial and deformed structures coexist, the initial structure is drawn last, so as to "cover" the deformed structure (effect visible on POSTSCRIPT screens and plotters, in particular);
- non-support node markers of the initial structure are drawn in "normal" size, those of the deformed structure are halved in size;
- node numbers and the numbers, section-type names and material-type names of elements are displayed on the initial structure.

These provisions give the initial structure a certain emphasis or "priority" of representation (PRIO INI option).

Deciding to grant priority to the deformed structure representation (PRIO DEF option) results in:

- crossing of line types of active initial elements, and of deformed elements;
- swapping of the drawing order of the initial and deformed structures;
- crossing of initial and displaced non-support node marker sizes;
- transfer of node numbers and the numbers, section-type names and material-type names of relevant elements to the deformed structure.

In both cases, the inactive initial elements are represented by dashed lines.

Please note that when the initial or deformed structure only is made visible and without priority, no "portable" attributes are visible.

The following attribute details are displayed on the initial structure, and are not transferable to the deformed structure:

- support node markers;
- support reference frames and placed support--type names;
- direction markers of elements and yz element reference frames.

This option is disabled in the absence of a deformed structure.

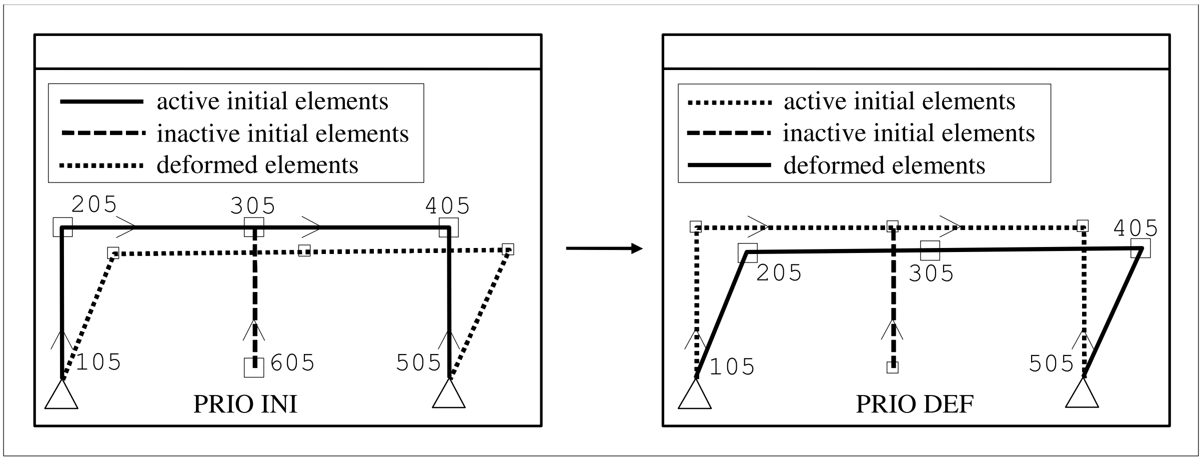


Figure 7.10 - Priority given to the initial structure, and then to the deformed structure

LEGE STRU

This option manages the display of the structure legend that attests to the supposed presence of the display elements, and their status.

When the initial structure is declared priority (PRIO INI option), the type of line used for the representation is:

- solid for active initial elements (1);
- dashed for inactive initial elements (2);
- dotted for deformed active elements (3).

When the deformed structure takes priority (see PRIO DEF option), the line types (1) and (3) are crossed.

Five line colors distinguish the elements according to their class (rigid, articulations, biarticulations, standard and beams). Articulation elements are represented by squares marking their end nodes.

The contents of the structure legend adapts automatically, when changing the selection on the STRU INI / STRU ACT / STRU TOT, STRU DEF or PRIO INI / PRIO DEF option.

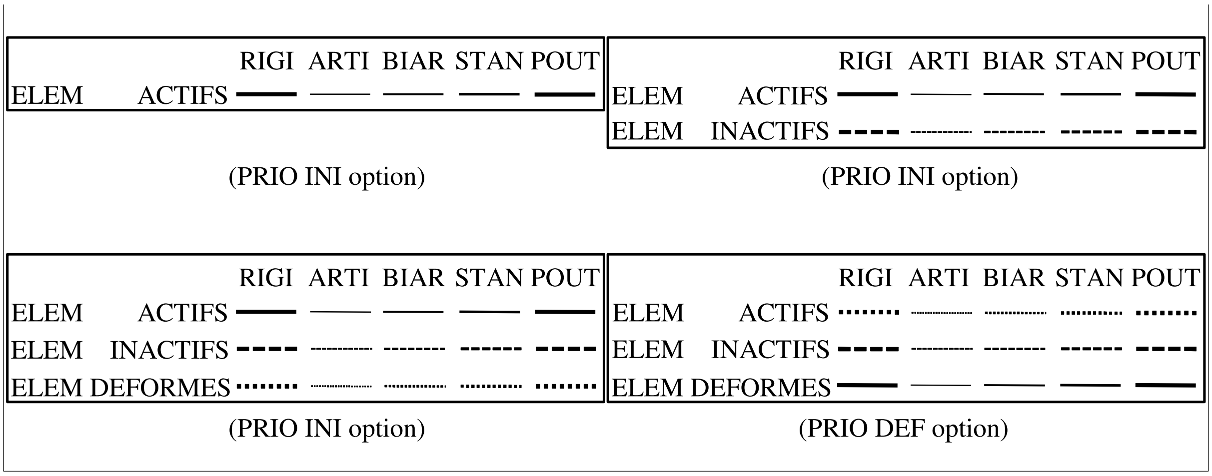


Figure 7.11 - Some possible aspects of the structure legend

MARQ NOE

This option manages the simultaneous display of:

- squares marking non-support nodes of initial and deformed structures;
- triangles marking support nodes of the initial structure.

Only the nodes connected to the visible elements of the current substructure (square markers) and the visible support nodes of the current substructure (triangle markers) are concerned.

The PRIO INI / PRIO DEF option determines the relative sizes of initial and deformed structure node markers.

NUME NOE

This option manages the display of initial and deformed structure node numbers.

The structure given priority by the PRIO INI / PRIO DEF option receives this numbering.

Only the nodes with positive numbers (which excludes the nodes generated by eccentricity) connected to the visible elements of the current substructure are concerned.

Each number can occupy 16 position-types, revolving around the projected node and examined in descending order of priority. Collisions are detected and avoided wherever possible; numbers causing inevitable collisions are displayed automatically.

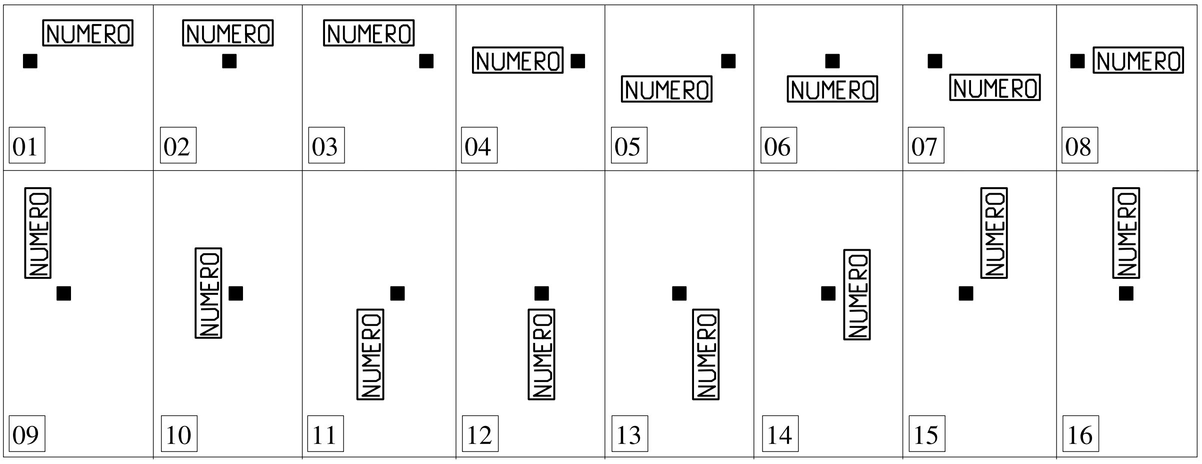


Figure 7.12 - Node number position-types

Collisions between node numbers and the numbers, section-type names or material-type names of elements are not managed (possibility to display them on separate drawings).

REPE APP

This option manages the simultaneous display of the projected support reference frames and names of support-types placed on the visible current initial substructure support nodes.

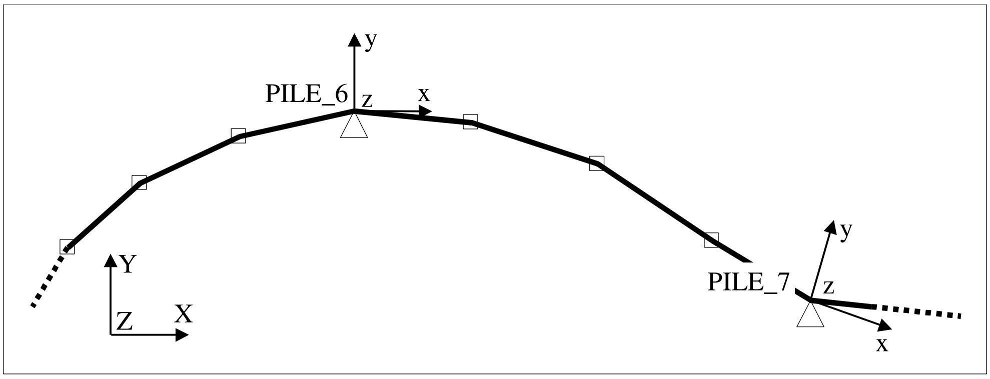


Figure 7.13 - Support reference frames and support-type names (top view)

HABI ELE / NUME ELE / SECT ELE / MATE ELE

This option manages the display of the numbers (NUME ELE), section-type names (SECT ELE) or material-type names (MATE ELE) of elements of the initial or deformed structure.

The structure given priority by the PRIO INI / PRIO DEF option receives these attributes.

In addition to the visible articulations (ad hoc), the only elements of the current substructure concerned are those for which the projected length of the visible part is greater than a minimum set at 3 mm on the screen, and at 2 or 2.5 mm on the plotter; these valuesare multiplied by 1.732 in magnifying glass mode (see ZOOMER action).

For rigid elements, only positive numbers (which excludes items generated by eccentricity) are displayed and there are no section-types or material-types assigned.

For articulation elements, the numbers are displayed in the same way as nodes and the names of any articulation--types placed replace the section-type names; no material-types are assigned.

For the beam elements, section-type names are obtained by concatenation of ‘P_’ and the beam number concerned.

Each number can occupy 48 position-types, or 16 position-types (analogous to those of Figure 7.12) successively revolving around a point situated at one quarter, half, and three quarters of the visible part of each projected element, following its direction; the numbers follow any inclination of the horizontal elements; these positions are examined in a decreasing order of priority which is influenced by the presence of several elements of the same incidence.

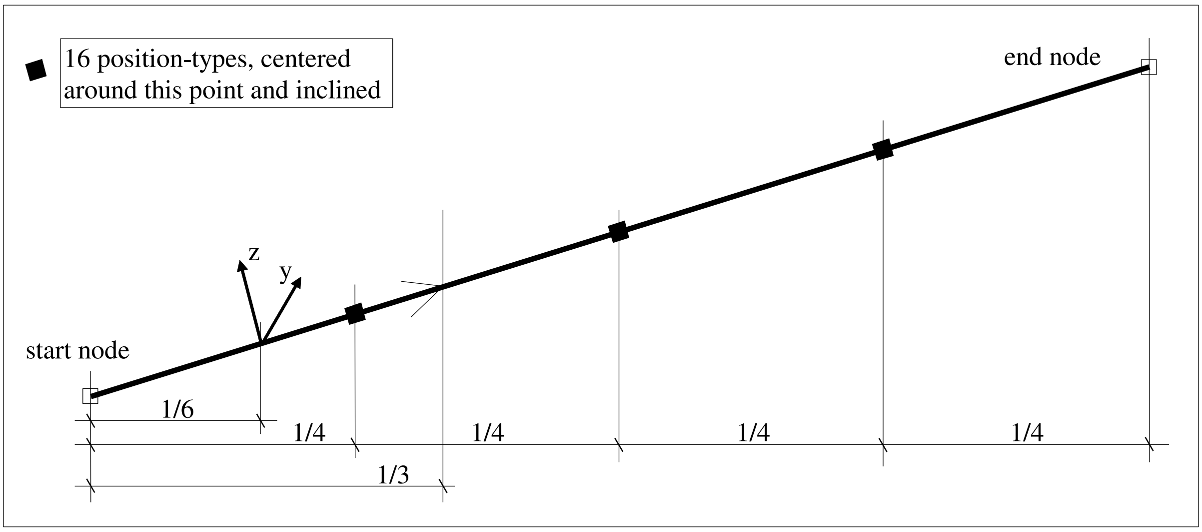


Figure 7.14 - Full element, position-types for attributes, direction, local yz reference frame

Collisions are detected and avoided wherever possible; attributes causing inevitable collisions are displayed automatically.

Collisions between these attributes and the node numbers are not managed (possibility to display them on separate drawings).

SENS ELE

This option manages the display of direction marking arrows (start node => end node) of non-articulation elements of the initial structure.

The only elements of the current substructure concerned are those for which the projected length of the visible part is greater than a fixed minimum (see HABI ELE / NUME ELE / SECT ELE / MATE ELE option).

The points of the arrows are positioned in the first third of the parts of visible elements, following their direction (Figure 7.14).

REPE ELE

This option manages the display of local projected yz reference frames of non-articulation elements of the initial structure.

All elements of the current substructure located in the field of view are concerned, including those whose projection is ad hoc or treated as such.

For elements that are not projected ad hoc, the reference frame origins are positioned in the first sixth of their visible parts, following their direction (Figure 7.14).

Oriented mean lines of elements are the x axes of their direct local reference frames.

RESU ELE

This option manages the display of a group of "curves" representative of results at the ends of elements selected by the RES module, via the ETUDES and EFFETS options in its main menu, and their sub-options; it is disabled in the absence of results.

Only active non-articulation elements of the current substructure are concerned, with at least one end belonging to the designated area of study.

Representation technique

The values are displayed at the the ends of the elements, along optional extension lines (see TRAI RAP option) "orthogonal" to their unmarked mean line (continuous fine white line on screen or black on the plotter); the corresponding points are connected by straight lines to form "curve segments".

The orthogonality of extension lines is only effective if you work in same scales mode (ECHE EGA option); otherwise, they are simply parallel.

The scale factor transforms all data valuesin lengths to be shown in the 2D reference frame of the projected model; this provision creates a coupling between the model geometry and the lengths of the extension lines, via the global scale factors.

Positive valuesare conventionally represented above a horizontal element from left => right.

On graphics hardware that manages color, each individual curve (or envelope) is shown in solid lines and distinguished by its color; on monochrome hardware, four types (solid, dashed, dotted, mixed) are used and two line thicknesses.

All extension lines are solid and fine, white on screen and black on the plotter.

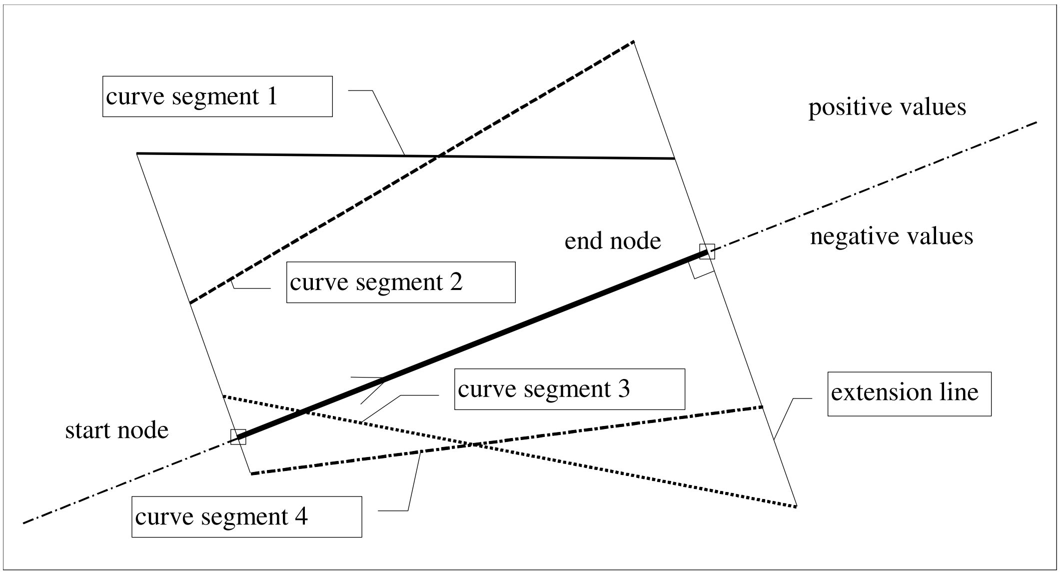


Figure 7.15 - Four curve segments for a full element (same scales)

Main conventions

The model results and a deformed structure can not be displayed at the same time.

When the DESS OPT option is inactive and the RESU ELE option is enabled, the results (not shown) are included in the full window.

We can not simultaneously represent more than 8 "single"curves or 8 "envelope" curves (pairs of curves with the same attributes).

An "identification label" of the same color may be affixed on each curve (single or envelope) of which at least a part is visible; these labels can be transferred to a "results legend"; it is also possible to display (on curves or legend) only the beginning of the labels, which has a fixed "maximum effective length."

In full window mode, auto reframing occurs on the nodes, elements and results displayed.

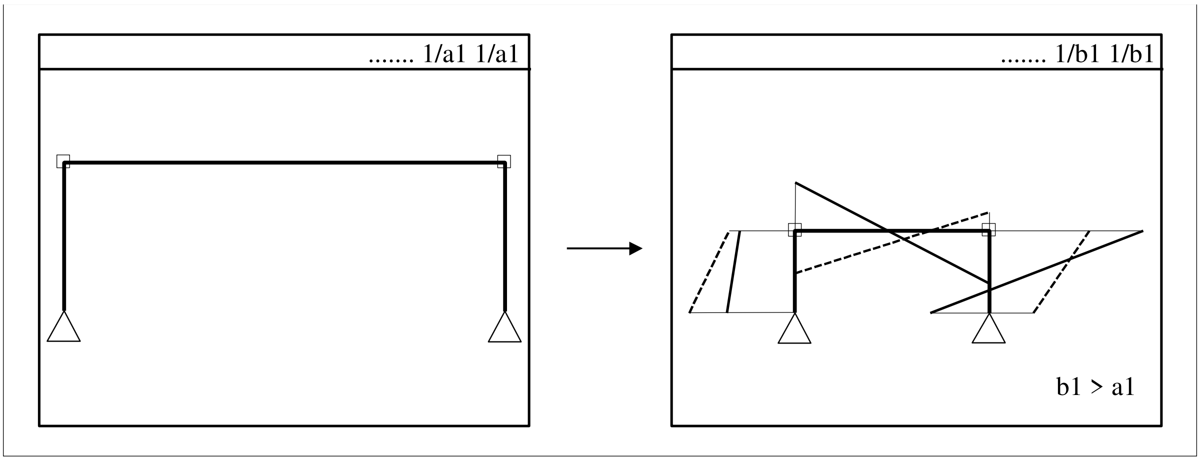


Figure 7.16 - Addition of results in full window mode (same scales)

In partial window mode, framing and scale factors are not altered.

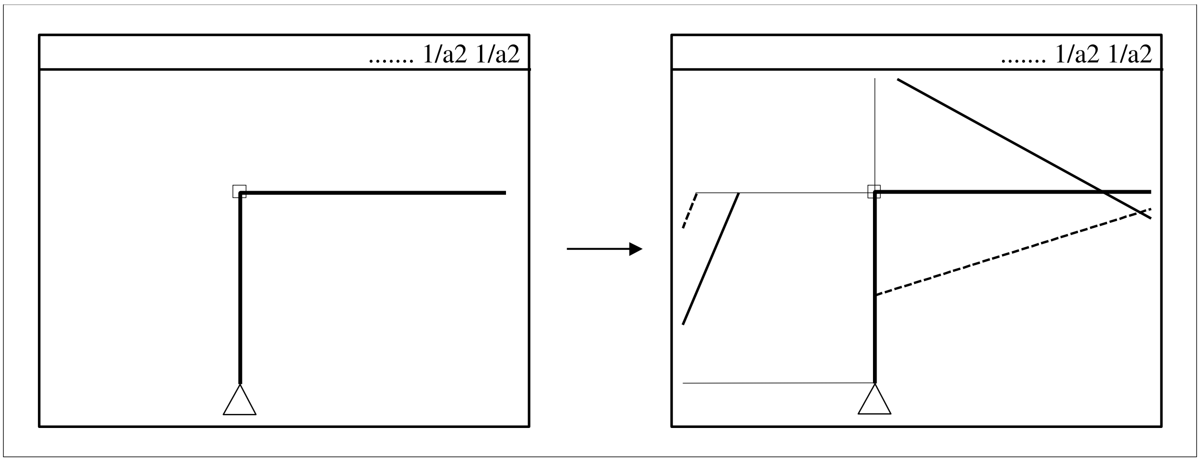


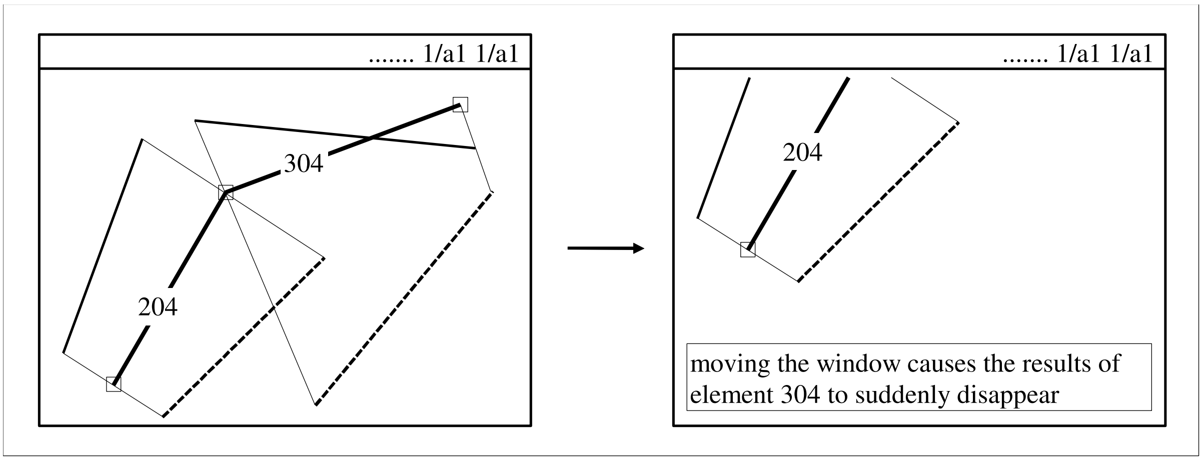
Figure 7.17 - Addition of results in partial window mode (same scales)

Visibility criteria

When all the results valuesat the beginning and the end of an element are zero, the curve segments that would be combined with its mean line are not represented.

Curve segments are not taken into account for elements for which the actual total projected length is less than 1 mm.

When the visible part of a projected element is less than 0.025 mm (on screen or on the plotter, value not affected by the magnifying effect of the ZOOMER action), curve segments are not drawn. It follows in particular that a zoom on a curve portion not including items related to it can cause its temporary "disappearance".



moving the window causes the results of element 304 to suddenly disappear

Figure 7.18 - Main results visibility criteria

TRAI RAP

This option manages the display of all extension lines on curves representative of the results at the ends of elements.

It is disabled when the results are absent or not displayed.

LABL LEG / LABL RES

This option manages the transfer of identification labels of curves from the results legend (LABL LEG default option) to the curves (LABL RES), or vice versa.

The algorithm for displaying labels on curves tests the curve segment visibility, operates a dispersion of labels, if possible on several elements, and tries to keep them in the field of view. However, the results legend is displayed globally, without any presumption of non-visibility of certain curves.

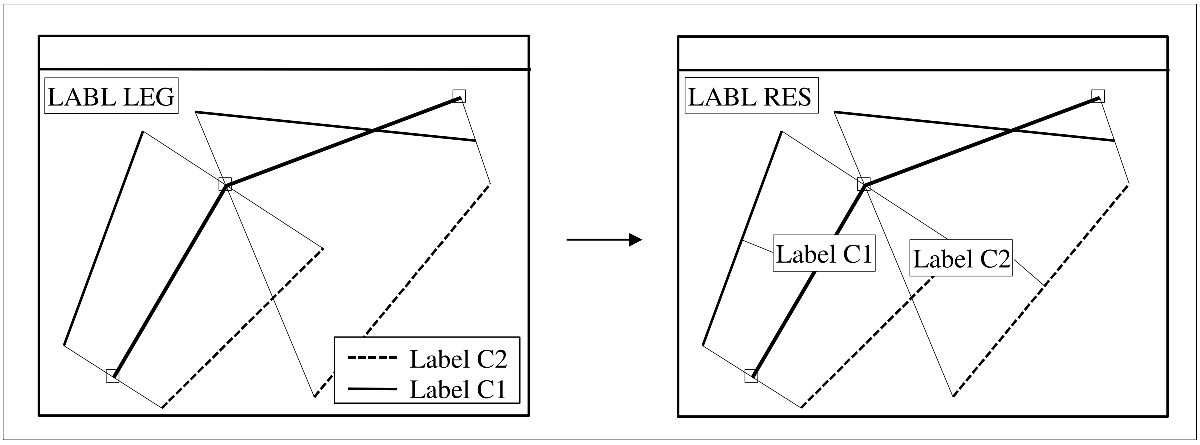


Figure 7.19 - Results legend dispersion on curves

IMPR TOT / IMPR 4X4 / IMPR ZON / IMPR ECH / IMPR NOR

This option allows selection of the plotter reproduction mode for all or part of the drawing on the screen.

Five print options exist that can be run cyclically in the order above; one of them is always active.

When the plotter display space is full, printed drawings are called "non-standard" and can be single or quadruple. They then include the title area, the model drawing area and a subtitle area, to which is added an additional optional print subtitle. The scale factors are calculated or imposed and there can be no overflow or error.

The "normalized" printed drawings are enclosed in a rectangular frame (consisting of "A4 formats'"stacked in width and/or height and delimited by fold crosses), plus a title block in A4 format containing a header, the title and subtitles. The plotter display space is not filled and if its size permits, drawings can be stacked vertically, then horizontally, until full. The scale factors are imposed and in the case of overflow, an error message is issued and the relevant printing is canceled.

The printing mode selected by default is non-standard for "reduced" format plotters, and standard for "large" format plotters (above A0). However, we can apply the standard method to reduced format plotters (A4 formats then take fictitious sizes), or a non-standard method for large format plotters (not recommended).

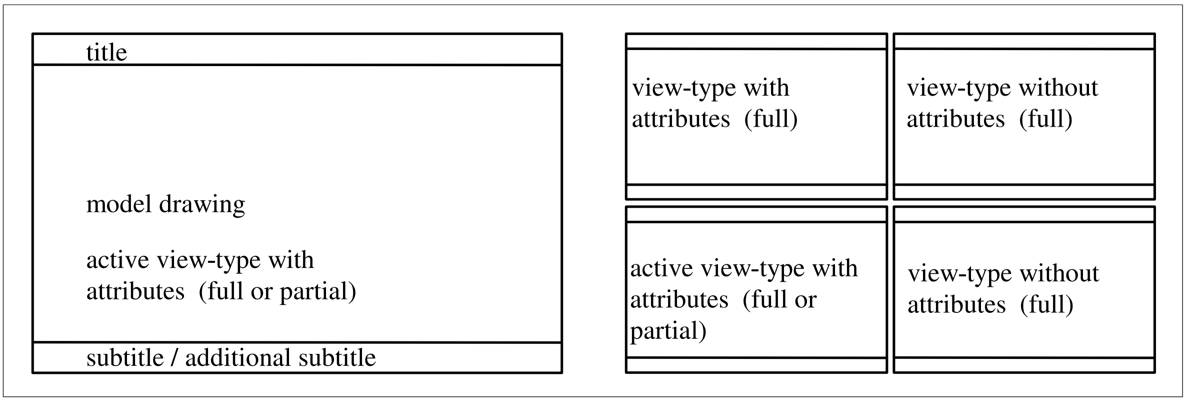


Figure 7.20 - Non-standard printing, single and quadruple

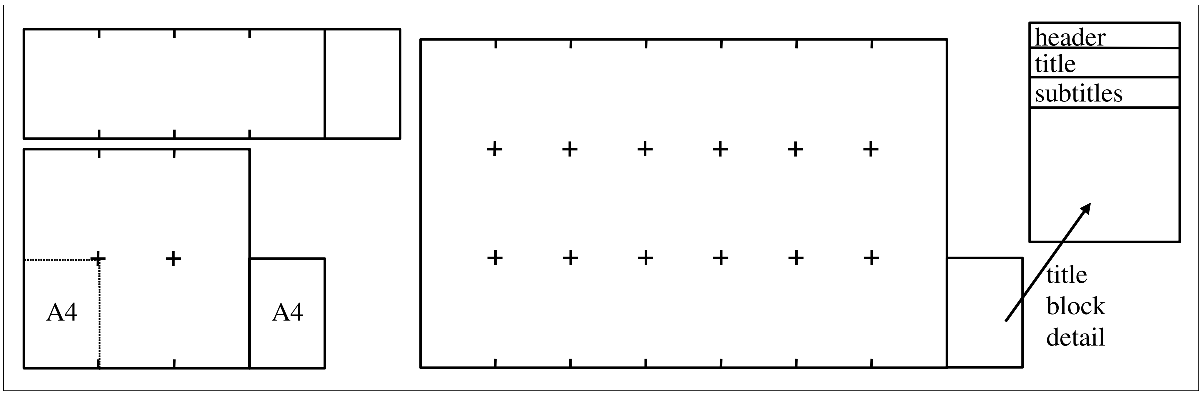


Figure 7.21 - Stacked standard printing

The table below summarizes the main conventions attached to different print modes, and how they are implemented.

Printing mode	Part of screen printed	Plotter saturation	Scale factors
IMPR TOT, non-standard single full printing *	Full active view-type, in 1X1 or 4X4 mode	Yes	Calculated, same or different, depending on ECHE EGA / ECHE DIF option
IMPR 4X4, non-standard quadruple full printing	Active view-type other full view-types, in 1X1 mode, full screen in 4X4 mode	Yes	Calculated, same or different, depending on ECHE EGA / ECHE DIF option
IMPR ZON, area printing, single non-standard	Rectangular part capture of the active view-type, in 1X1 or 4X4 mode	Yes	Calculated, same or different, depending on ECHE EGA / ECHE DIF option
IMPR ECH, partial printing, single non-standard, with imposed scales	Rectangular part of the active view-type in 1X1 or 4X4 mode, designated by its center; its size depends on the scale factors provided	Yes	Same or different, depending on the valuesprovided
IMPR NOR, standard printing with imposed scales *	Rectangular part capture of the active view-type, in 1X1 or 4X4 mode	No, stacked drawings	Same or different, depending on the valuesprovided

Table 7.3 – Printing modes

`(*) default options`

A single print can be obtained when using the single or quad screen sharing mode (see AFFI 1X1 / AFFI 4X4 option), and vice versa.

Depending on the screen sharing mode, the mode of accounting for scale factors (see ECHE EGA / ECHE DIF option), the current window type (full or partial), the width/height ratio of the plotter display space and the print mode selected, the number of possibilities is very large.

The figures below show the appearance of drawings printed in two very common use cases (width/height ratio of the plotter display space is set at 1.414).

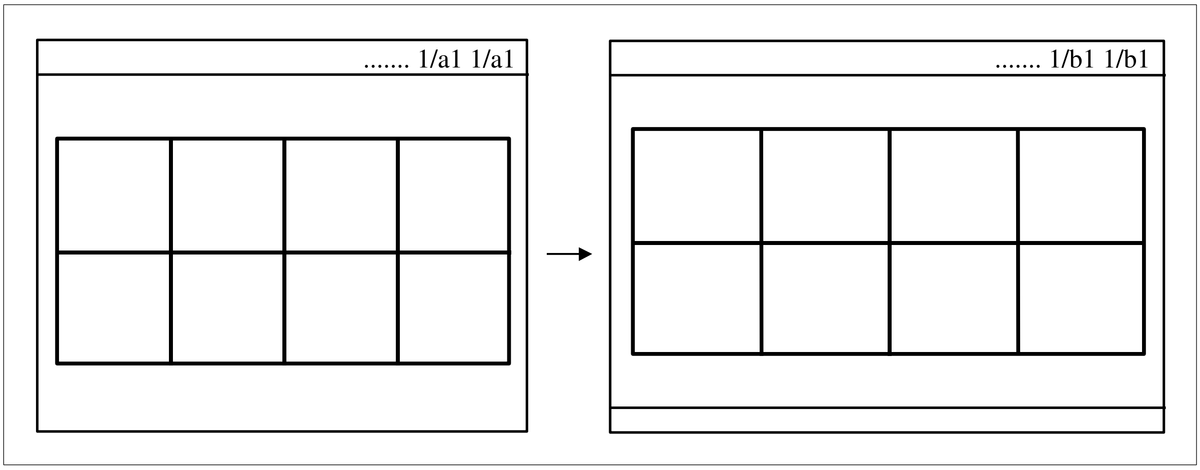


Figure 7.22 - Same scales full window full printing

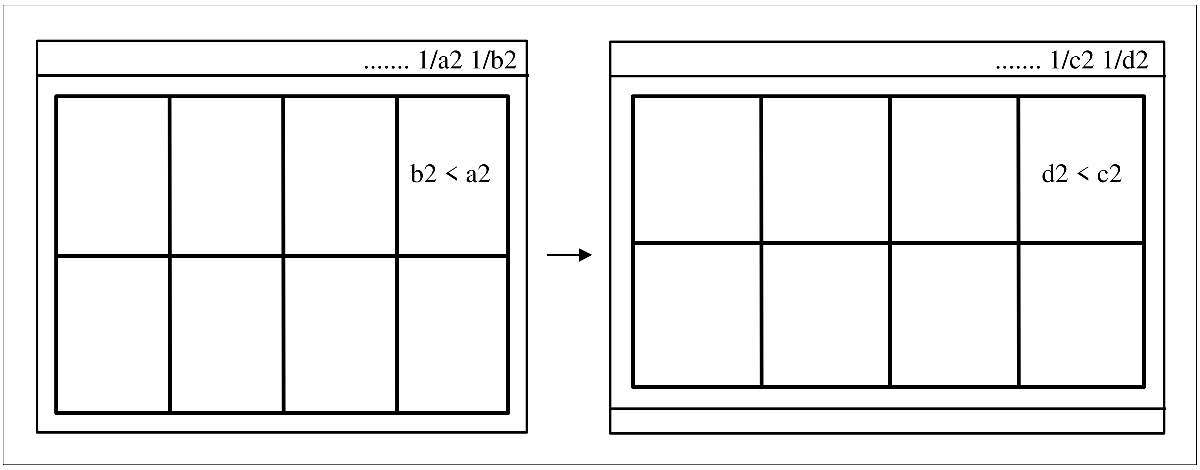


Figure 7.23 - Different scales full window full printing

Full printing preceded by zooming in on an area is equivalent to printing the area.

DESS OPT

This option manages the global and individual display, on screen, of the following attribute details:

- structure legend (LEGE STR option);
- node markers (MARQ NOE option);
- node numbers (NUME NOE option);
- support reference frames and support-type names (REPE APP option);
- element numbers, section-type names or material-type names (NUME ELE / SECT ELE / MATE ELE option);
- element direction markers (SENS ELE option);
- element yz reference frames (REPE ELE option);
- element end results, with identification labels (RESU ELE option).

When inactive, the requested details remain invisible and attribute option changes are stored, but not reflected in the display.

When it becomes active, the requested details are displayed simultaneously and the attribute option changes are reflected individually in the display.

Please note that the requested attribute details are taken into account for printing (IMPRIMER action), whether the DESS OPT option is active or not.

The VISEUR, AFFI 1X1 / AFFI 4X4, ECHE EGA / ECHE DIF, STRU INI / STRU ACT / STRU TOT, STRU DEF and PRIO INI / PRIO DEF options are not controlled by the DESS OPT command box.

Recommendations for use

Disable the DESS OPT option and choose the appropriate element attributes option before selecting a substructure (CHOISIR action) if working in full window and if the display risks being slow and unclear.

Disable the DESS OPT option before making the choice of options for printing if the drawing on the screen risks being overloaded.

7.2 - ACTIONS

DECOUPER

This action causes the current initial substructure to be cut by a plane perpendicular to the projection plane.

Its effects are cumulative and can be combined with those of the CHOISIR action, and the continued presence of the current substructure is ensured.

Any deformed structures adapt to changes in the current substructure.

The trace of the cut plane on the projection plane is denoted by two points in the drawing area of the model; the third point entered (necessarily outside the cut trace) denotes the half-space to be "deleted"; the nodes in the half-space and the elements attached to it are removed from the current substructure.

In full window mode, auto reframing occurs on the nodes, elements and results displayed.

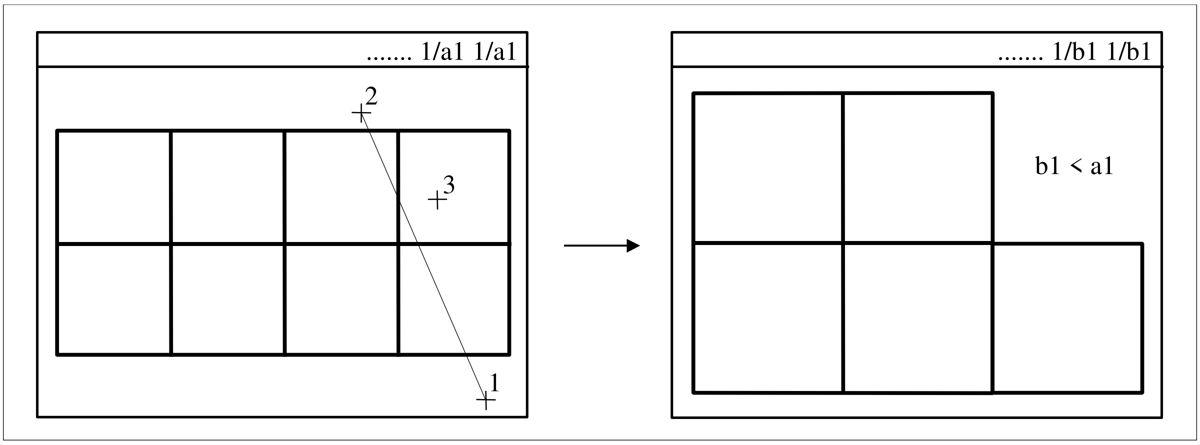


Figure 7.24 - Cutting current substructure in a full window (same scales)

In partial window mode, framing and scale factors are not altered.

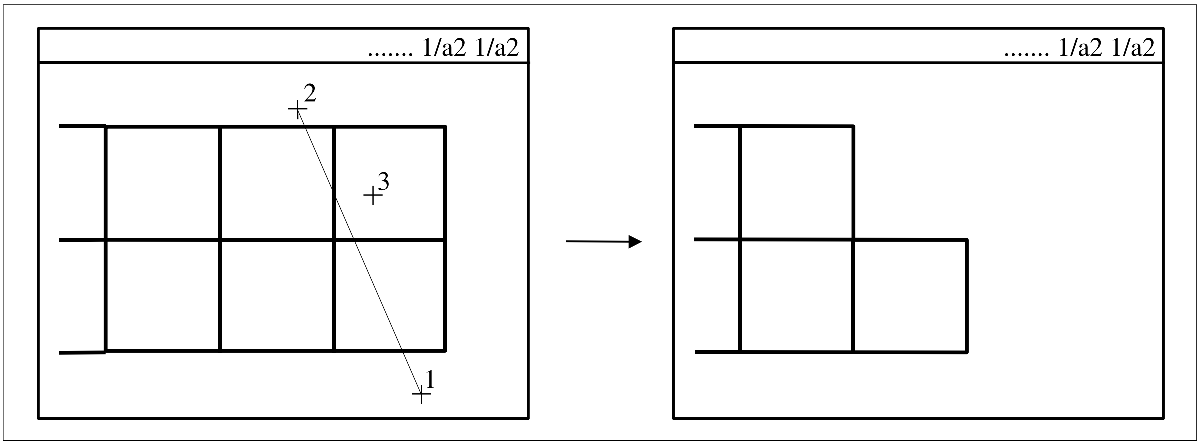


Figure 7.25 - Cutting current substructure in a partial window (same scales)

CHOISIR

This action (re)defines the current substructure through the selection of a group of elements, or modifies it by adding or removing a group of elements (entered as data).

If modified, its effects are cumulative and can be combined with those of the CHOISIR action, and the continued presence of the current substructure is ensured.

In full window mode, auto reframing occurs on the nodes, elements and results displayed; in partial window mode, framing and scale factors are not altered.

Any deformed structures adapt to changes in the current substructure.

Selection mode

The element attribute mode (see HABI ELE / NUME ELE / SECT ELE / MATE ELE option) determines the selection mode (whether active or inactive):

- by class, if the attribute is disabled (HABI ELE option)
- by range of numbers, limits inclusive (NUME ELE option);
- by particle attached to section-type (SECT ELE option) or material-type names (MATE ELE option).

The structure legend (see LEGE STR option) recalls the abbreviations used to denote the five standard element classes (RIGI, ARTI, BIAR, STAN, POUT).

We add the RESU class denoting the elements on whose ends the results are provided.

Prefixing operator

The selection can be preceded by an operator:

- +, if adding elements to the current substructure;
- –, if removing elements;
- *, if only keeping those current substructure elements that match the new selection criterion.

When the operator is absent, the current substructure is redefined.

Remarks, recommendations for use

The four main selection modes can be combined freely, by changing the HABI ELE / NUME ELE / SECT ELE / MATE ELE option.

If necessary, use the PROJETER and DECOUPER actions, alternating with the CHOISIR action.

‘P_i’ type particles can be used to select certain beams, in SECT ELE attribute mode.

The selection particles for section-type and material-type names are not controlled for verification of the lexical name writing conventions; so they can start or end with '_', begin with a number or contain only numbers.

Examples

The terms of each selection are separated by at least one space.

Selection	Type	Meaning
POUT	By class	The current substructure is redefined and includes beam elements
+ STAN	By class	Standard elements are added to the current substructure
* RESU	By class	Only current substructure elements that have results are retained
100 500	By range of numbers	The current substructure is redefined and includes elements with numbers between 100 and 500
+ 750 1000	By range of numbers	Elements are added to the current substructure with numbers between 750 and 1000
– 400 400	By range of numbers	Element 400 is removed from the current substructure
– 21	By particle in section-type names	Current substructure elements with section-type names containing particle 21 are removed
+ BET	By particle in material-type names	Elements with material-type names containing particle BET are added to the current substructure

Table 7.4 - Some element selection examples

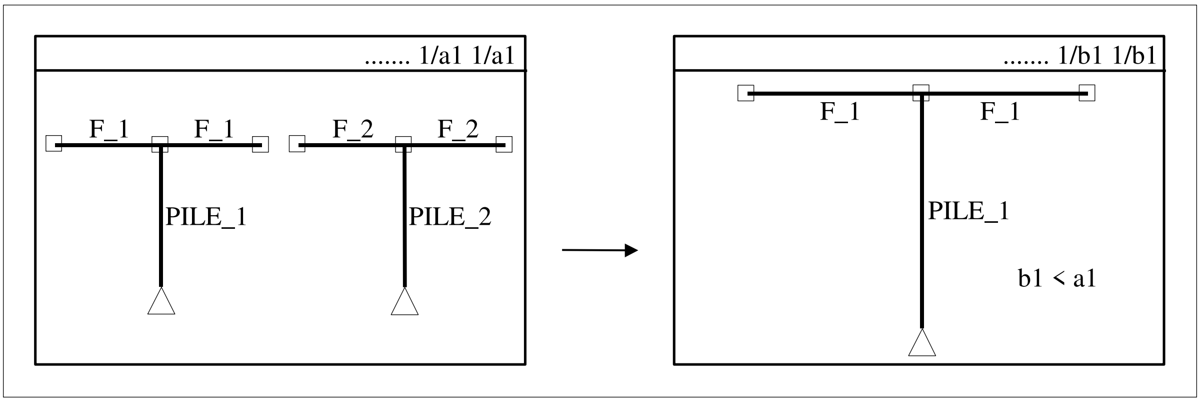


Figure 7.26 - SECT ELE option, selection effect - 2 in full window (same scales)

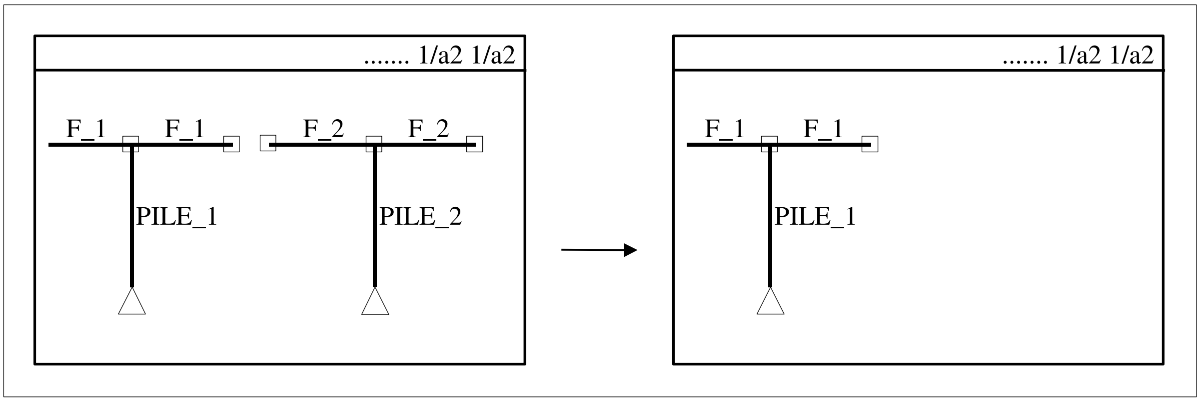


Figure 7.27 - SECT ELE option, selection effect - 2 in partial window (same scales)

RECOLLER

This action cancels the effects of all cuts (DECOUPER action) and element selection (CHOISIR action) since the start or the last RECOLLER action; the full initial structure is restored as the current substructure (default option).

It has no effect if the structure has not been subject to any cuts or element selection.

In full window mode, auto reframing occurs on the nodes, elements and results displayed.

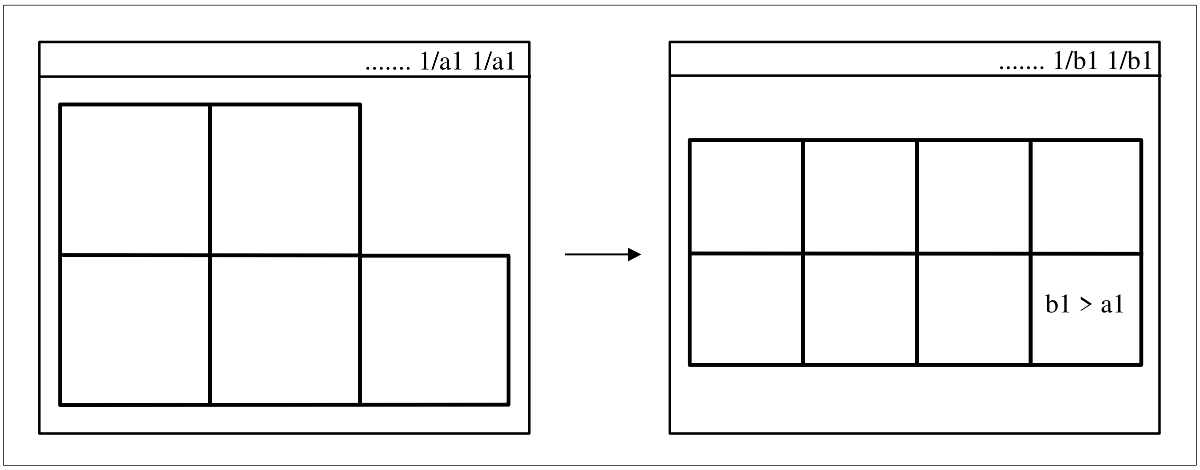


Figure 7.28 - Recombining in a full window (same scales)

In partial window mode, framing and scale factors are not altered.

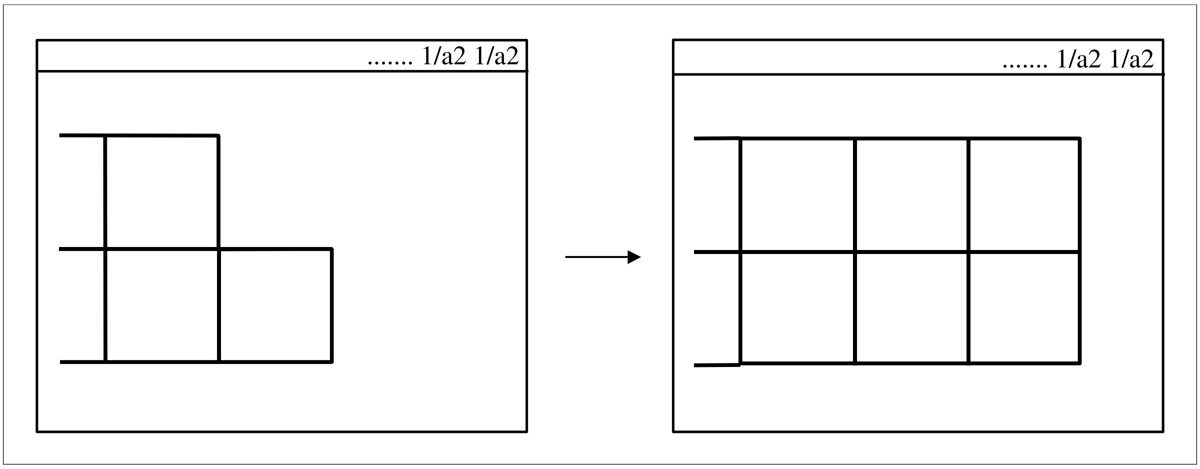


Figure 7.29 - Recombining in a partial window (same scales)

ZOOMER

This action causes the capture of a rectangular part of the model drawing area, which will become, after any adjustment, the new on screen viewing window or printing window in full mode (see IMPR TOT option).

It also causes the eventual switching from full window mode to partial window mode.

In same scales mode (ECHE EGA option), the captured part is enlarged symmetrically in width or height, to obtain new equal scale factors, and fills the model drawing area in width or height.

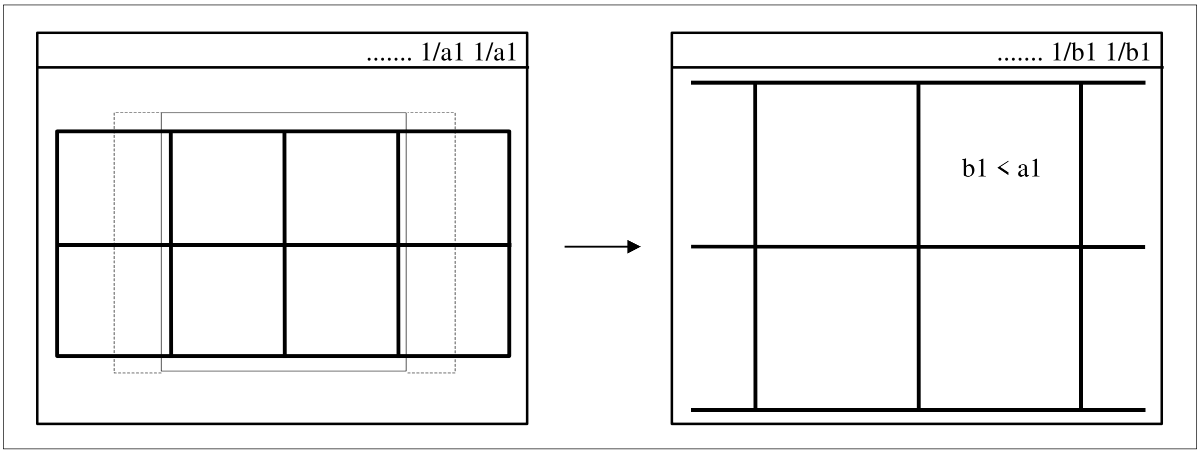


Figure 7.30 - Zooming in same scales mode

In different scales mode (ECHE DIF option), the captured part is retained and fills the model drawing area in width and height, with generally different recalculated scale factors.

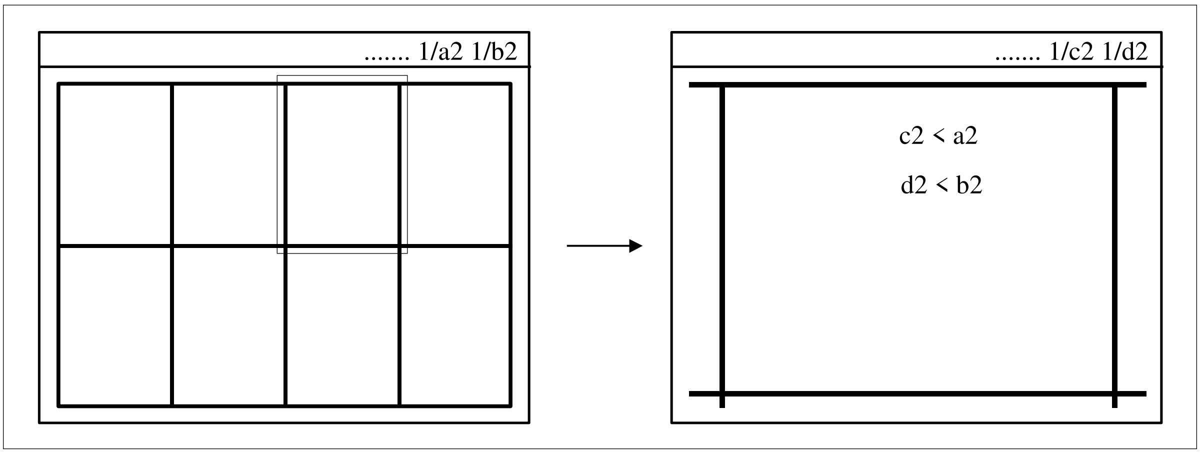


Figure 7.31 - Zooming in different scales mode

Magnifying glass mode

When the captured rectangle has a width and a height of less than 2% of those of the model drawing area, zoom is supposed "ad hoc" and the "normal" mode (selected by default) gives way to "magnifying glass" mode, or vice versa.

In "magnifying glass" mode, most nominal line thicknesses are doubled and most attribute details (symbols, and texts outside the legends) have their nominal sizes multiplied by 1.732; the scale factors and framing are not altered.

Please note that the magnifying glass mode also applies to printouts (see IMPRIMER action).

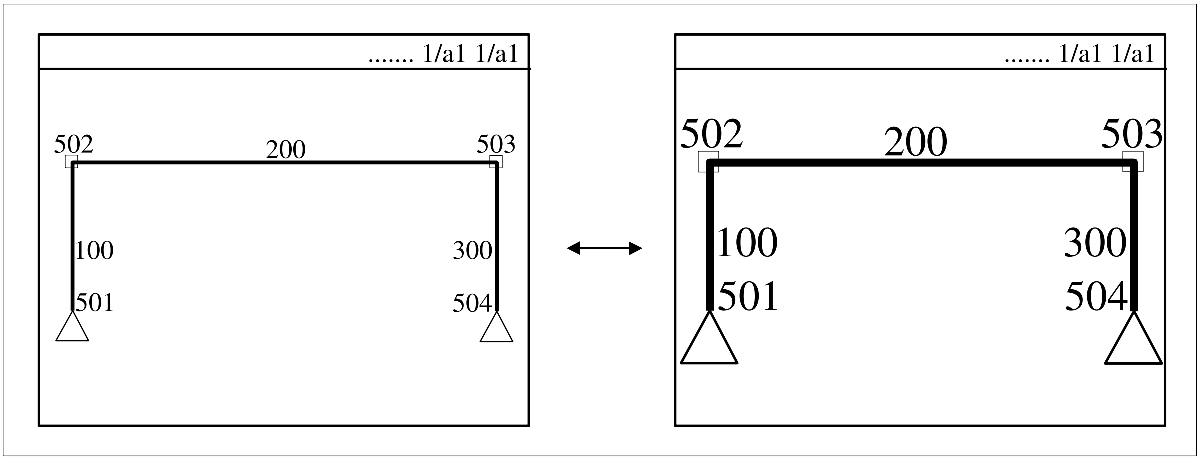


Figure 7.32 - Some magnifying mode effects

RESITUER

This action allows a magnification or shrinkage coefficient entered in data to be applied to the on screen model drawing; framing is altered proportionally but the central point of the drawing (middle of the reticle) is retained.

It also causes the eventual switching from full window mode to partial window mode.

The structure appears to get farther away if the coefficient is greater than 1.0, or to get closer if is less than 1.0. Its lower bound is set to 0.1, there is no upper bound.

The "apparent" effect is zero if the coefficient is 1.0, but this option can be used to "force the passage" to partial window mode and prevent subsequent automatic reframing.

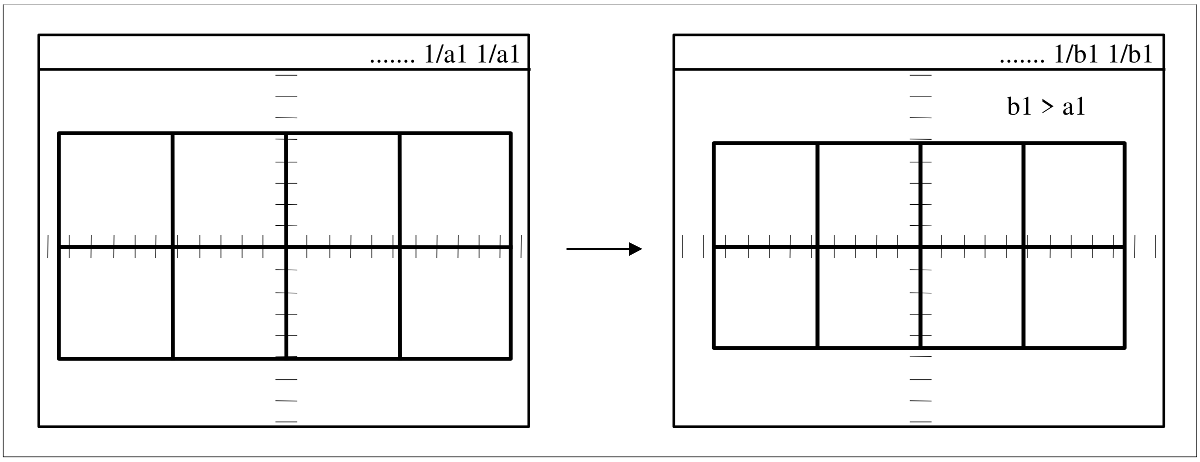


Figure 7.33 - Shrinkage applied in same scales mode (coefficient 1.10)

FIXE ECH

This option allows two scale factors entered in data (width and height) to be applied to the on screen model drawing; framing is modified accordingly but the central point of the drawing (middle of the reticle) is retained.

It also causes the eventual switching from full window mode to partial window mode.

The lower bound of a scale factor is set to 0.05, there is no upper bound.

In same scales mode (ECHE EGA option), if two different values are entered, only the highest is taken into account as a scale factor, to be applied in width and height; the second is set aside and applied only if a switch to different scales mode occurs (ECHE DIF option).

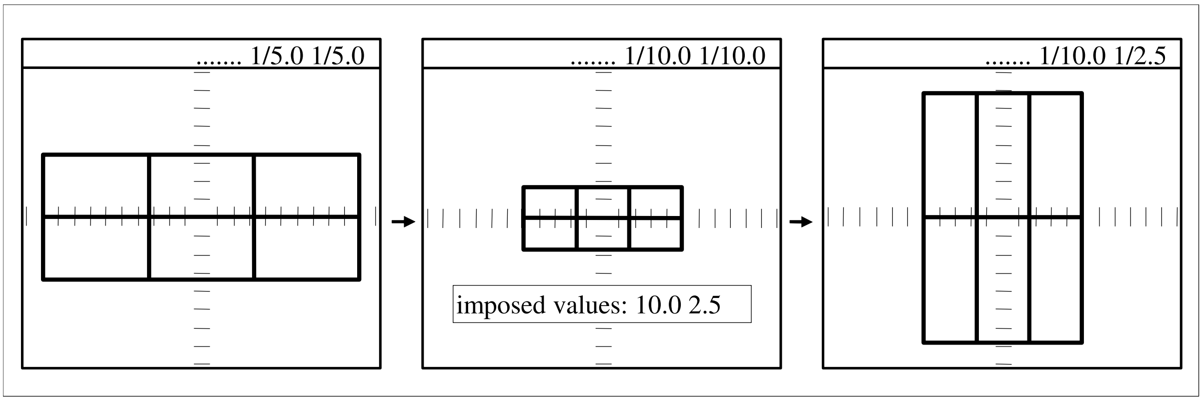


Figure 7.34 - Imposed factors in same scales, switching to different scales

In different scales mode (ECHE DIF option), both valuesare taken into account, in width and height, in their order of entry.

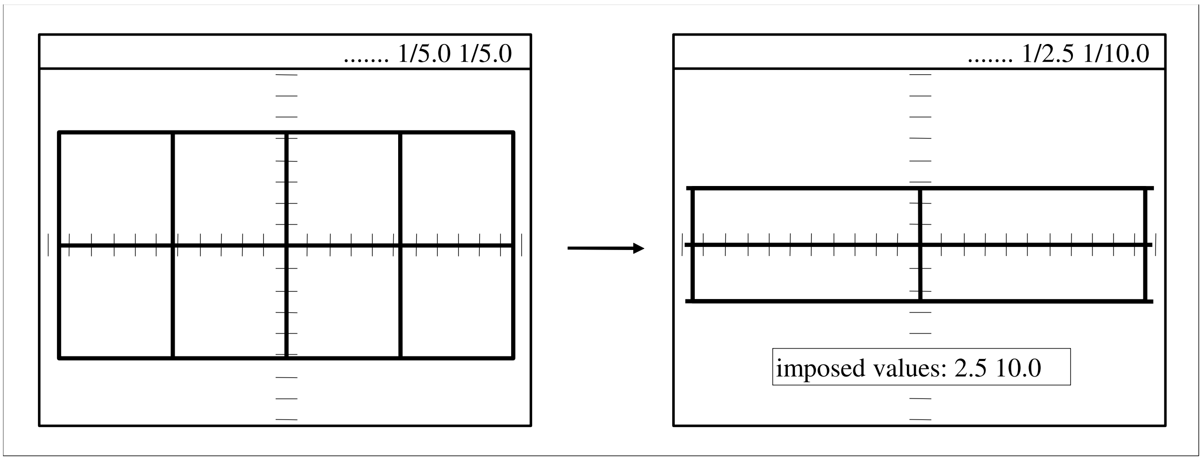


Figure 7.35 - Imposed factors in different scales

DEPLACER

This action causes a translation of the display window; the center of the reticle is virtually displaced to a captured point from the model drawing area; the scale factors are retained.

It also causes the eventual switching from full window mode to partial window mode.

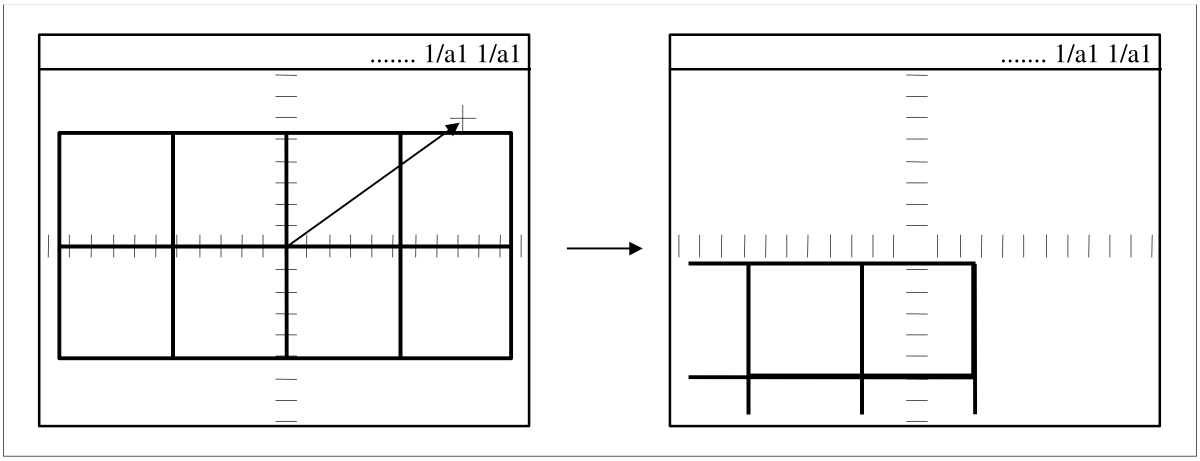


Figure 7.36 - Moving window in same scales mode

CHERCHER

This action causes a translation of the display window; the center of the reticle is virtually displaced to a node or the middle of an element, for which the number is provided, preceded by the keyword NOEUD or ELEMENT; the scale factors are retained.

It also causes the eventual switching from full window mode to partial window mode.

The node or localized element can remain invisible if it does not belong to the drawn model.

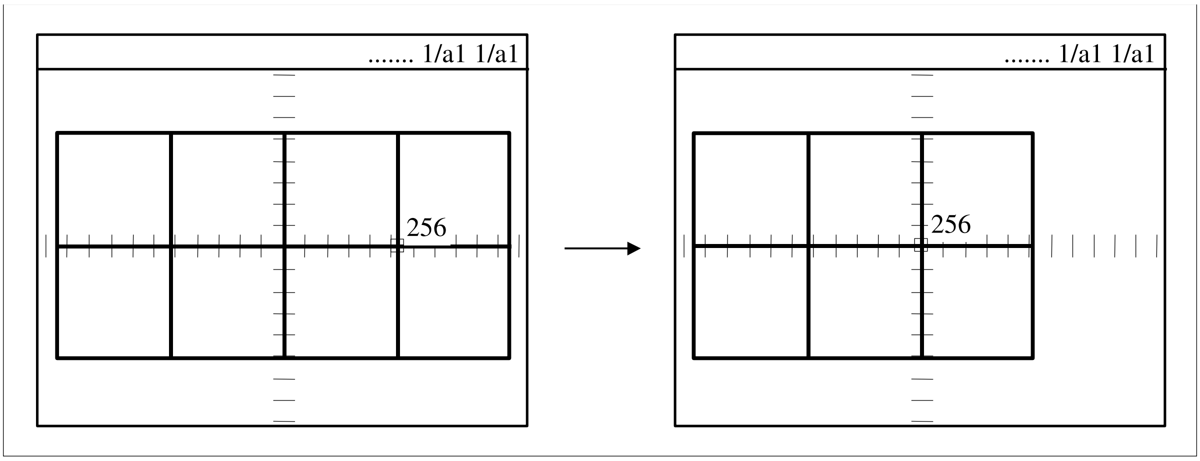


Figure 7.37 - Searching for NODE 256 in same scales mode

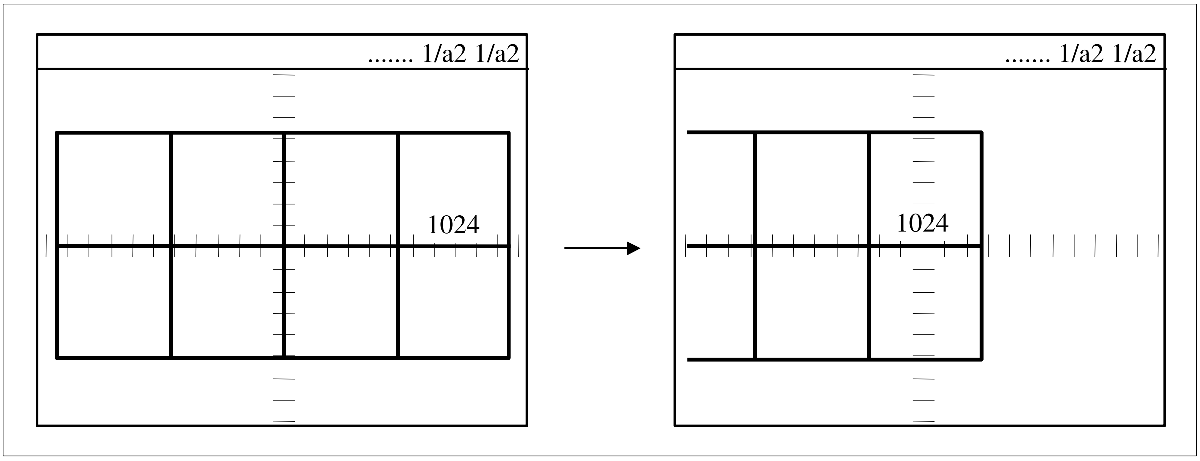


Figure 7.38 - Searching for ELEMENT 1024 in same scales mode

TOURNER

This action causes rotation of the model in its projection plane, the parameters are defined by three points P1, P2 and P3 captured in the drawing area; P1 is the center of rotation; P2-P1-P3 is the oriented rotation angle.

It can be useful, in particular, to put a perspective "back in line" before printing (on a roll plotter).

Several rotations can be done in sequence whose effects are cumulative; only the change in the projection plane of the active view-type (see XY>XZ>YZ or PROJETER action) cancels their effects.

We keep track of 10 rotations at most, made since the last change in the projection plane, in order to restore the effects when changing the display mode (see AFFI 1X1 / AFFI 4X4 option).

In full window mode, auto reframing occurs on the nodes, elements and results displayed.

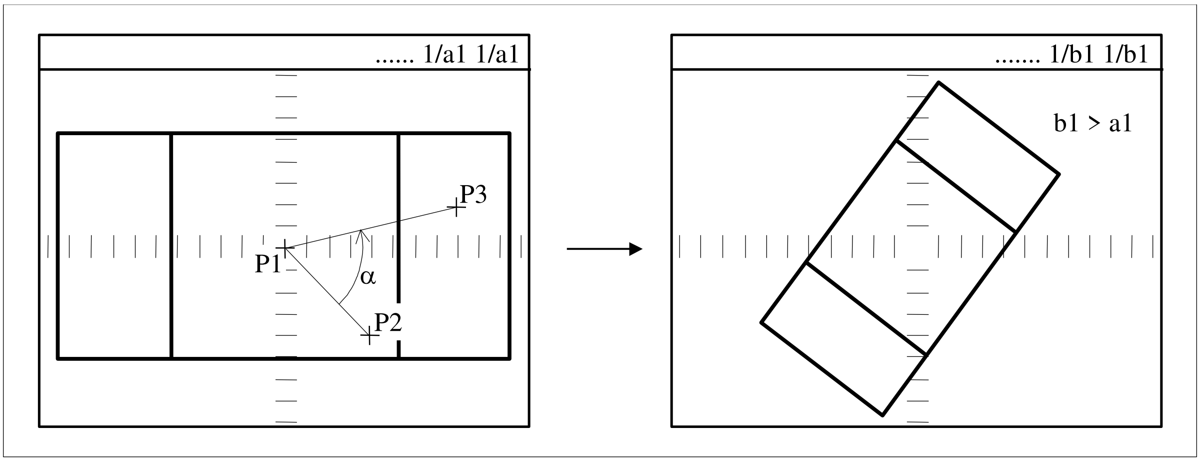


Figure 7.39 - Rotation around the center of the reticle, full window (same scales)

In partial window mode, framing and scale factors are not altered.

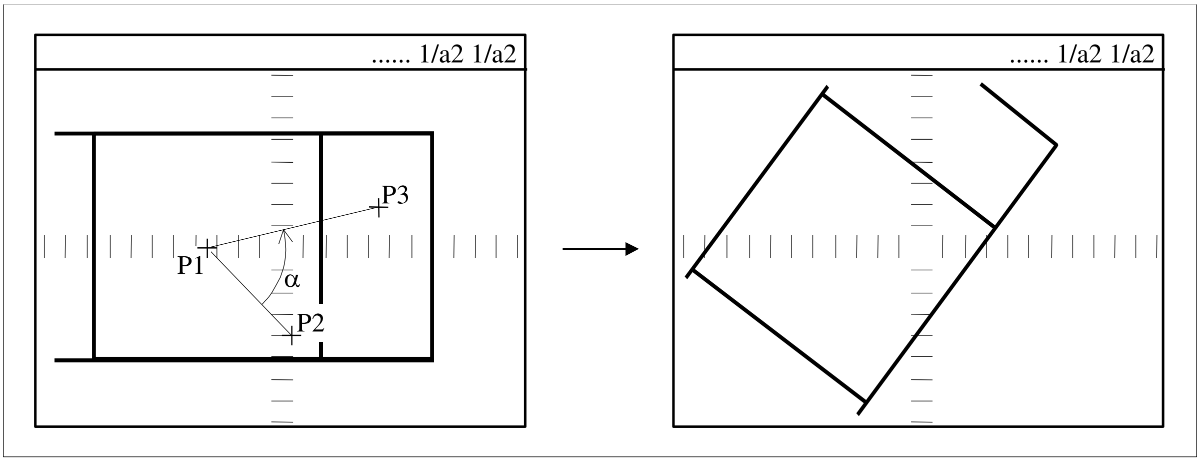


Figure 7.40 - Rotation around any point, partial window (same scales)

RECADRER

This action causes reframing of the drawing on the nodes, elements and results displayed, so as to make best use of the available space; the scale factors are recalculated accordingly; the safety edge allows entry of most attribute details, without clipping.

It also causes the eventual switching from partial window to full window mode.

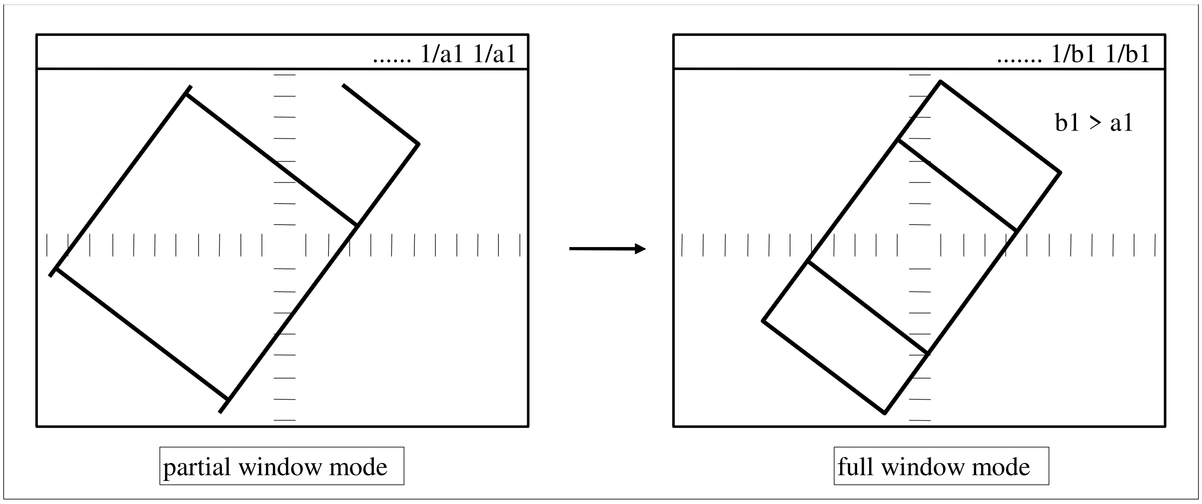


Figure 7.41 - Reframing after rotation, in same scales mode

AMPL DEF

This action changes the current value of the deformed structure displacement amplifying coefficient, for the active view-type only; it has no immediate effect if the deformed structure is not to be displayed, or is disabled in its absence.

This coefficient simultaneously multiplies the three displacement valuesof the active nodes of the complete model and its continued presence is ensured.

It is set by default to 10% of the quotient of the maximum structure size (along the axes of the global reference frame: OX, OY and OZ for perspective, OX and OY for the top view, OX and OZ for elevation, or OY and OZ for the view from the right) by the maximum absolute value of displacement, and replaceable by any positive or zero value.

In full window mode when the deformed structure is requested (active STRU DEF option), auto reframing occurs on the nodes and elements displayed.

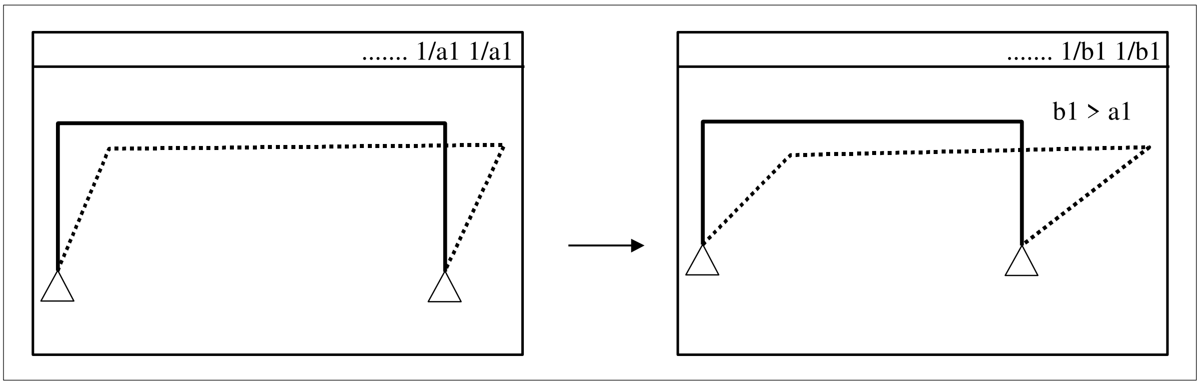


Figure 7.42 - Displacement amplification increased in full window

In full window mode, when the deformed structure is not requested, or in partial window mode, framing and scale factors are not altered.

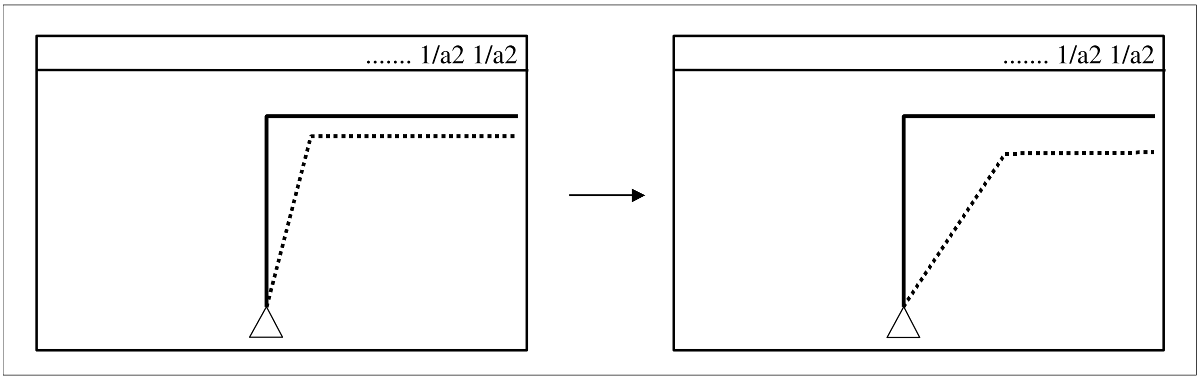


Figure 7.43 - Displacement amplification increased in partial window

Recommendations for use

The amplifying coefficients calculated by default for the four view-types are reduced to two different valuesat maximum. To apply other coefficients, alternately use the XY>XZ>YZ and AMPL DEF actions.

ECHE RES

This action changes the current value of the scale factor for element end results; it is disabled in the absence of results.

This scale factor is the length equivalent to the largest absolute result value, for a group of curves to be displayed simultaneously; its continued presence is ensured.

It is set by default to 3/8 of the maximum dimension of the structure (along the axes of the global reference frame), and replaceable by any value greater than 0.1.

In full window mode, when the results are requested and displayed (active RESU ELE and DESS OPT options), auto reframing occurs on the nodes and elements displayed.

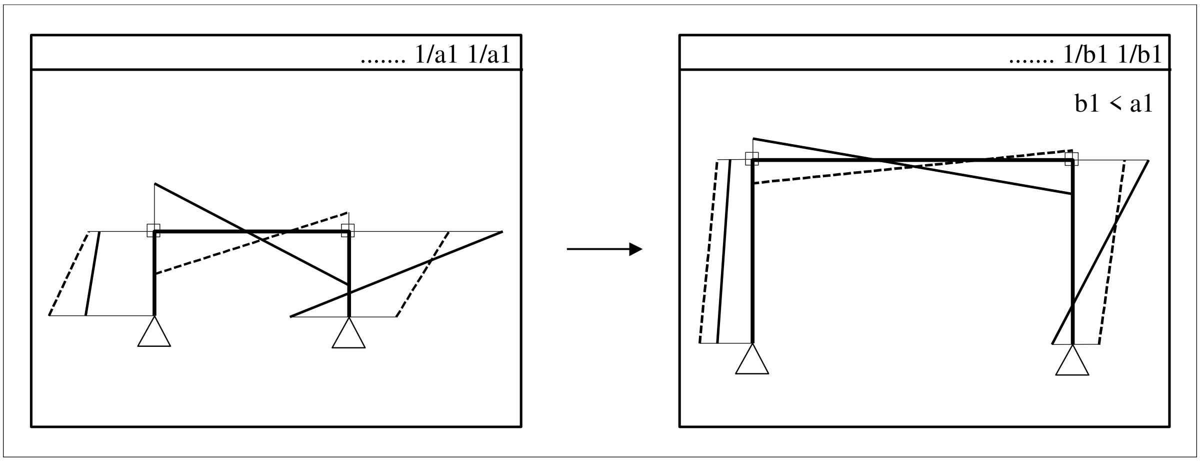


Figure 7.44 - Results scale factor reduction (full window, same scales)

In full window mode when the results are not visible, or in partial window mode, framing and scale factors are not altered.

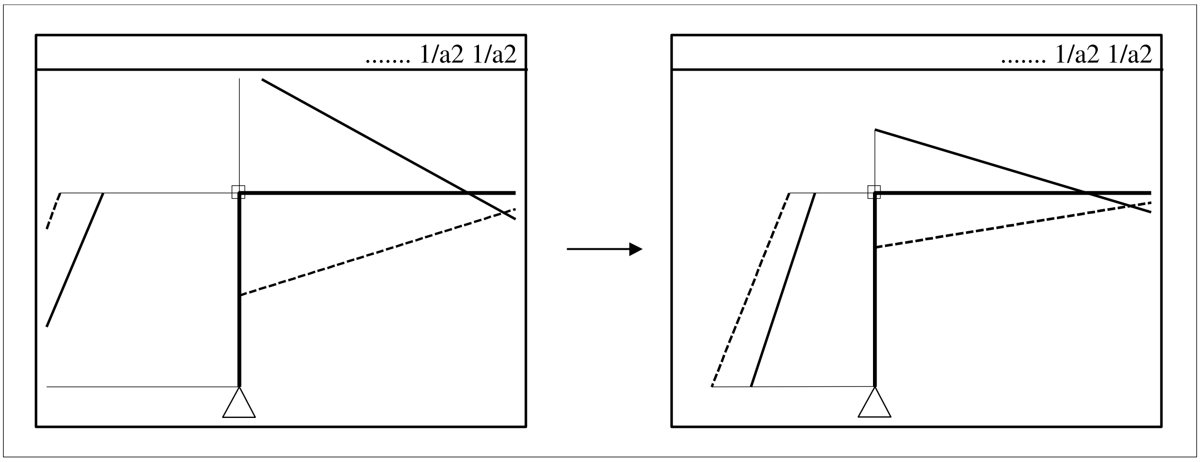


Figure 7.45 - Results scale factor reduction (partial window, same scales)

LONG LAB

This action changes the maximum effective length of curve identification labels; it has no immediate effect if results exist and are not displayed, or disabled in the absence of results.

By default, labels to be displayed (compacted and left justified) use at most 60 characters; this maximum effective length can be increased to any value between 0 and 80; zero leads to the disappearance of the labels.

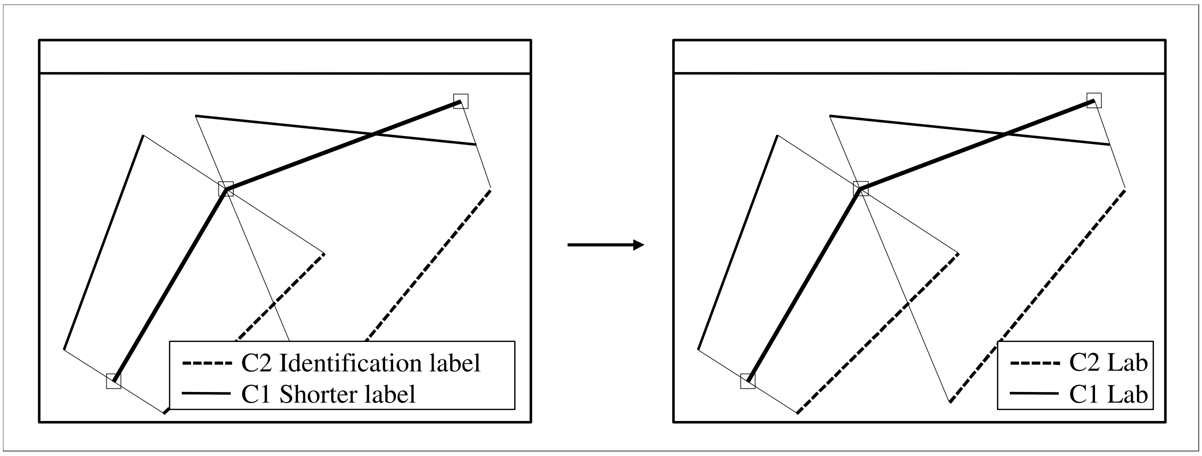


Figure 7.46 - The maximum effective length of legend labels passes from 25 to 6

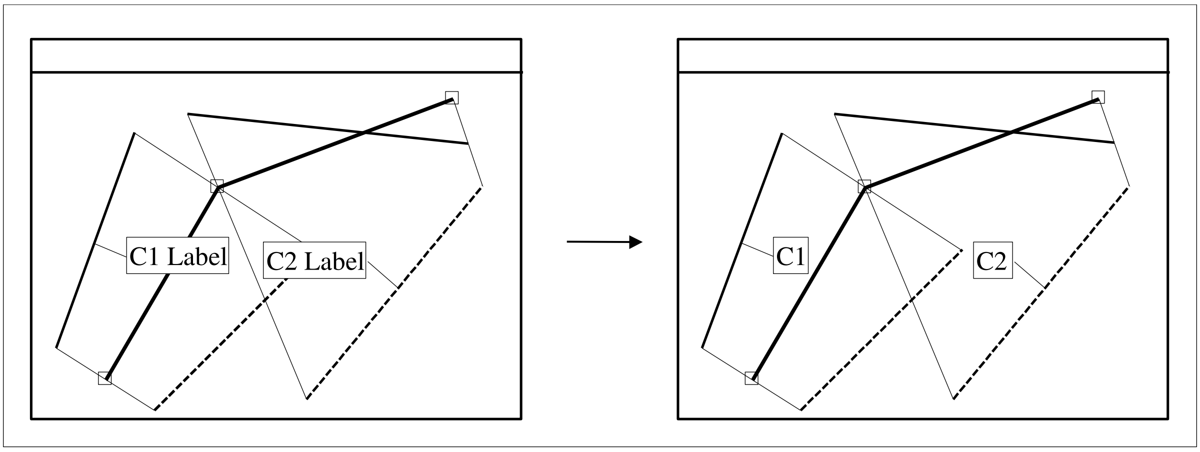


Figure 7.47 - The maximum effective length of curve labels passes from 8 to 2

POINTER

This action displays certain result valuesat the ends of elements on curves, or displacements on the nodes of the deformed structure, by repeated individual plotting; it is disabled when the curves or deformed structure are not displayed or absent.

Technique used

Centered at each point entered in the model drawing area, a "circle of proximity" is established, of radius 1.5 mm.

All curves are examined in order of introduction (see the results legend going from bottom to top), and point by point, in the order determined by the corresponding areas of study; displaced nodes are examined in the order of storage.

The selected point is the first curve segment end, or the first displaced node, encountered within the interior of the circle of proximity; it is identified by a circular marker and the result value ordisplacement values corresponding to it.

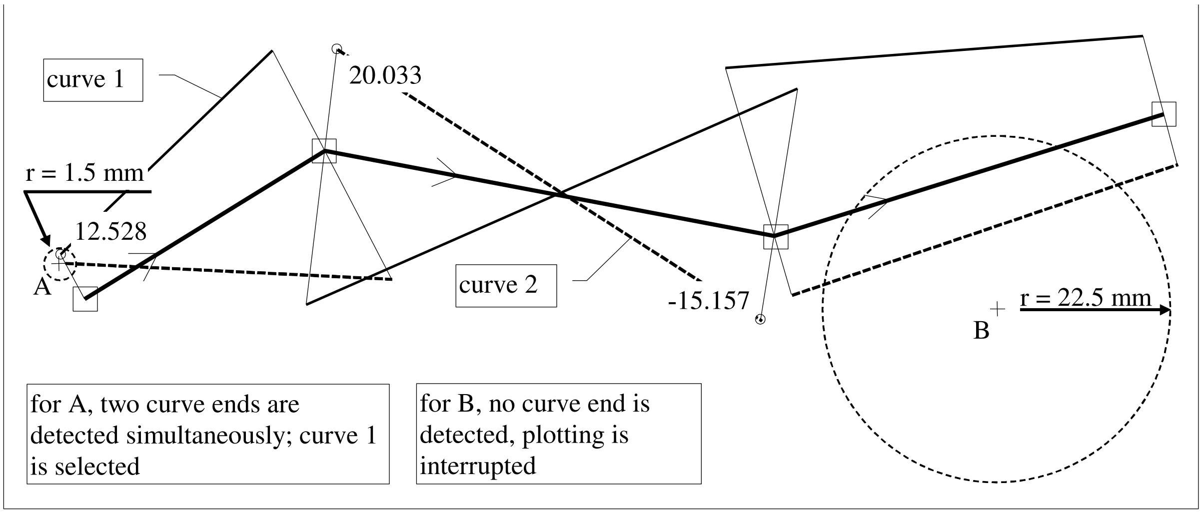
If no points are detected, the program renews the search by multiplying the radius of the circle by 15; if successful, no value is displayed but plotting can continue; otherwise, plotting is interrupted (exit possible by pointing "away from the curves or the deformed structure").

Plotting can also be interrupted by pressing the ESC key or its equivalent.

The continued presence of plotted valuesis ensured until the general eraser is used (see GOMMER V action).

Plotting values already displayedresults in their deletion (individual eraser).

On graphics hardware that manages color, markers and results valuesuse the colors of the curves to which they relate; markers and displacement valuesuse only white on the screen and black on the plotter.



for A, two curve ends are detected simultaneously; curve 1 is selected

for B, no curve end is detected, plotting is interrupted

Figure 7.48 - Plotting values on curves

The output formats used are as follows (limits given in absolute values):

- with exponent and four significant figures for valuesless than 0.1;
- fixed, with three decimals for values greater than or equal to 0.1 and less than 1.0E6;
- with exponent and six significant figures for values greater than or equal to 1.0E6.

Only values of displacements (unamplified) corresponding to "nonzero columns" are displayed, possibly separated by semicolons; the displacements legend, which indicates the meaning, also contains the amplifying coefficient applied, preceded by a star.

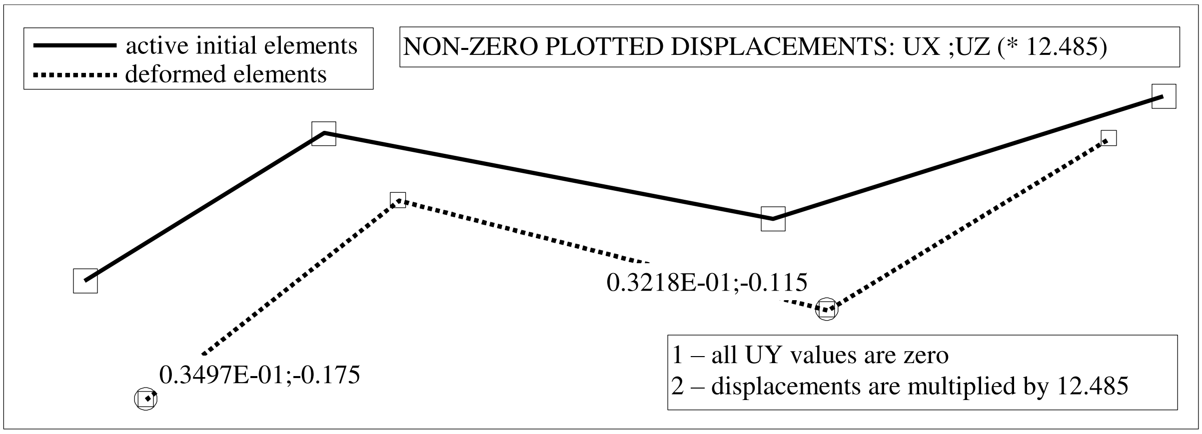


Figure 7.49 - Plotting node displacement on a deformed structure

According to the quadrants defined by the reticle and the frame of the drawing that the markers belong to, plotted valuescan occupy four relative position types, ensuring they remain in the field of view.

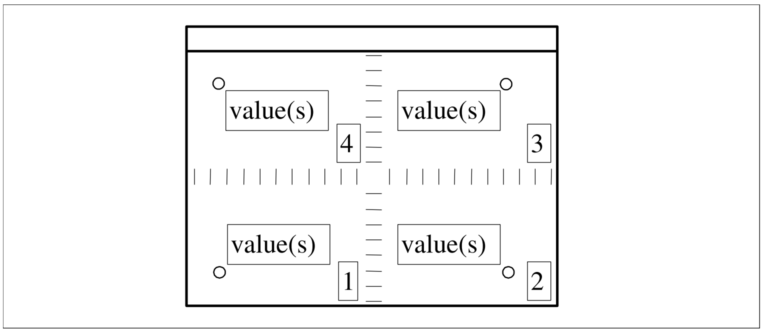


Figure 7.50 - Relative marker positions and plotted values

Recommendations for use

On screen, as plotted valuesregister on a black rectangular background, they overlap each other in the order of plotting when the density is high; we can make the hidden values reappear by double clicking. To remove traces of individually deleted values, refresh the display by any action.

When printed on hardware that manages overlaps (POSTSCRIPT plotters in particular), plotted valuesoverlap in a fixed, uncontrollable order.

GOMMER V

This action cancels the effects of all previous plotted valueson curves or node displacements; it is disabled when the curves or deformed structure are not displayed or absent. It has no effect if the curves or deformed structure are displayed without plotted values.

If the curves or deformed structure are visible, the display is refreshed, but the framing and scale factors are not altered.

XY>XZ>YZ

This action allows you to quickly switch from one to another of the four view-types, cyclically, in the order below:

PERSPECTIVE => VUE DE DESSUS => ELEVATION => VUE DE DROITE

The orientation angles of their local reference frames are set by default to the values below (they are modified by the PROJETER action):

View-type	θ_1	θ_2	θ_3	Projection plane
PERSPECTIVE	-45.0	0.0	-30.0	
TOP VIEW	0.0	0.0	-90.0	XOY, view in the direction opposite to OZ
ELEVATION	0.0	0.0	0.0	XOZ, view in the OY direction
VIEW FROM THE RIGHT	90.0	0.0	0.0	YOZ, view in the direction opposite to OX

Table 7.5 - Four view-types

This change in the projection plane is accompanied by a reframing on the nodes, elements and results displayed, and an eventual switch to full window mode.

In quad display mode, the new view-type becomes active and is located in the lower left corner; the other three view-types are swapped circularly.

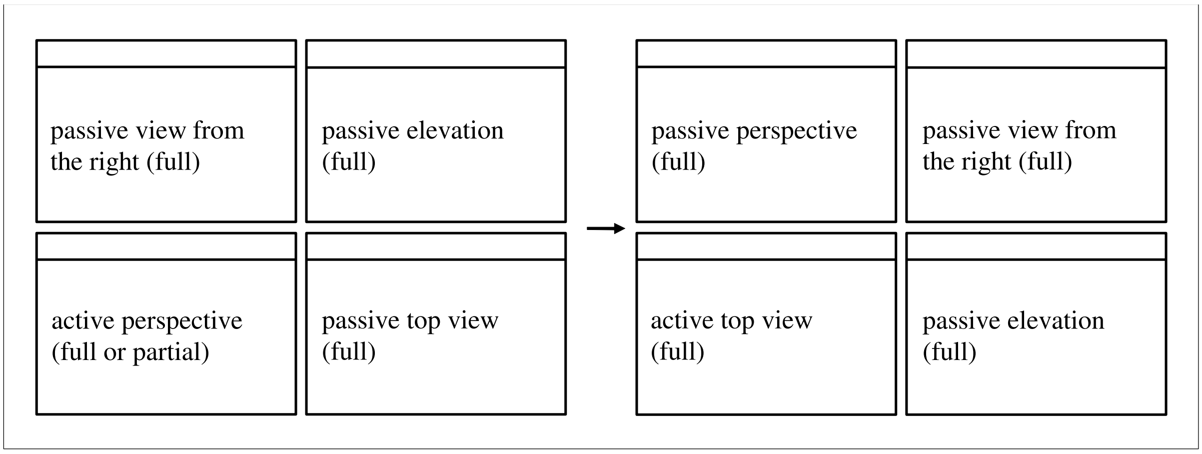


Figure 7.51 - Swapping of the four view-types, from their standard layout

Please note that the standard names of the four view-types may become inappropriate if the angles of orientation of their projection planes are changed.

PROJETER

This action allows the orientation angles of the projection plane of the active view-type to be changed.

Its effect is persistent and the modified view-type can be found again after passing through the other view-types (XY>XZ>YZ action).

It is accompanied by auto reframing on the nodes, elements and results displayed.

The model is supposed projected on the xOz plane of a reference frame, derived from the global reference frame by a triple rotation in accordance with the general conventions (see Figure 1.2); its Oy axis indicates the direction of observation.

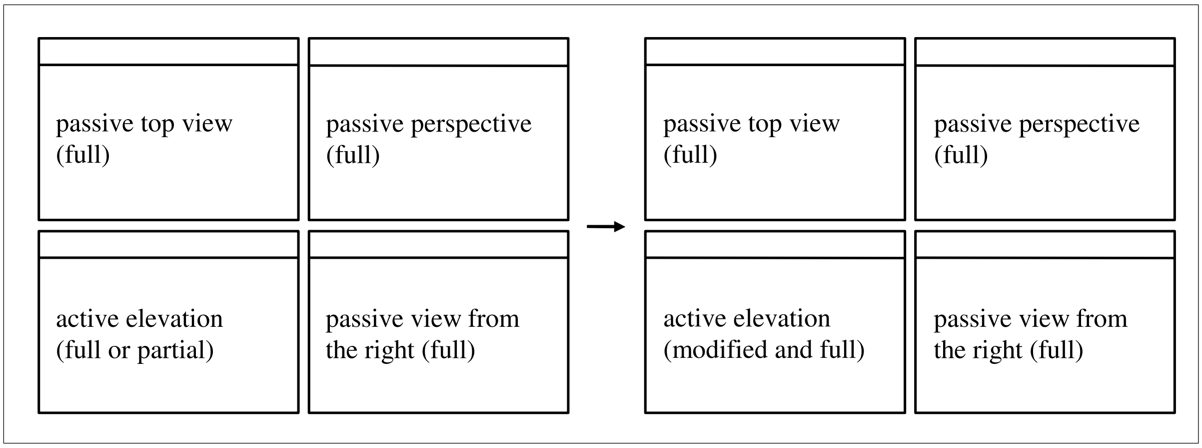


Figure 7.52 - Changing the elevation projection plane, in quad mode

CHGT IMP

This action allows the name of the current print file to be changed, or for drawings to be printed on multiple files to be sent; by default, they are stacked in a single file named: pcpres.tra (or pcpph2.tra for the reduced version).

IMPRIMER

This action allows all or part of the displayed drawing to be transferred to a file, depending on the print mode selected (see IMPR TOT / IMPR 4X4 / IMPR ZON / IMPR ECH / IMPR NOR option); Table 7.3 shows which part of the screen is printed.

All active options are taken into account, even if the screen does not display the attributes (inactive DESS OPT option), as well as the magnifying glass mode (ZOOMER action), if used.

The optional additional subtitle is entered in data, as well as any imposed scale factors (printing to scale or standard, IMPR ECH / IMPR NOR option); the lower bound of a scale factor is set to 0.05, there is no upper bound.

In standard print mode (IMPR NOR option), when the size of a drawing is greater than that of the plotter display space, it is not transferred, and the error message: "** IMPRESSION ARRETEE" is shown.

Chapter 8

Construction phases

Introduction

Annex A provides the full wording of the document referenced in this chapter under the condensed name: "IP 2", "BAEL 83" and "BPEL (99)".

In this chapter, the term "cable" refers to a prestressed cable, and the term "beam" refers to a spatial beam.

Functions of the PH3 module

This module is used to reproduce the successive CONSTRUCTION and CHARGEMENT PHASES of a construction by simulation, and to monitor any changes in its state of DEFORMATION and SOLLICITATION.

The VISCO-ELASTIQUES behavior of the materials over TEMPS, can be precisely taken into consideration if a construction HISTORIQUE is provided, otherwise it is approximated.

Thanks to the beam acquisition modules and the PH1 module, the geometric and mechanical characteristics of the structure model to be assembled and the instantaneous losses of tension in the cables are known.

The PH3 module must be provided with the details of the construction operations and the applied loadings, the selected calculation OPTIONS, any data that completes the definition of the model, and the directives for the EDITION and the ENREGISTREMENT of certain useful results.

At the origin, the structure is entirely deactivated and unsolicited. The rigidity of its supports and elements is gradually introduced as they are installed.

In each construction phase, the ACTIVE structure can comprise several substructures that are independent of one another, but whose STABILITE is necessary under the effects of the applied loads (instabilities due to isolated notes that are partially linked to stable substructures are automatically inhibited).

The numbers of the elements (mechanical entities) and the incidences (geometric entities) are initially in CONCORDANCE. If this concordance is temporarily suspended, certain elements may occupy incidences that are distinct from those that were assigned to them when they were defined. This measure allows POUSSAGE operations to be simulated.

The repercussion of the displacement and rotation of the nodes of the active structure on the free nodes connected to newly activated elements may be effective (in DEPLACEMENTS REPERCUTES mode) or not (in DEPLACEMENTS NON REPERCUTES mode). When only the displacements are concerned by this repercussion, and not the rotations, this is TRANSLATIONS REPERCUTEES mode.

The effects of the IMPLICITES loadings are automatically accumulated in the current state of the active structure. They are the result of the simulation of the following construction operations:

- installation of supports with jacking, removal or replacement of supports,
- installation, displacement or removal of form travelers,
- introduction of the dead weight of elements,
- installation of articulations with reestablishment of continuity or replacement of articulations,
- tensioning of elements or cables,
- removal of elements or loosening of cables,
- modification of the current date,
- redistribution of solicitations in a flat-rate manner.

The effects of the EXPLICITES loadings (grouped in CAS DE CHARGE) can be automatically accumulated in the current state of the active structure (in CUMUL mode), or not (in CUMUL SUSPENDU mode). This applies to:

- forces applied to nodes or elements,
- deformations of supports or distortion of elements,
- thermal loads on elements.

The ETU module can read the recorded displacements and solicitations (states of the structure, effects of a load case or a buckling mode), edit them or combine them together or with the compatible results produced by the ENV or DYN modules, or provided. The RES module can produce a graphical representation.

Terminology

The EFFET of a construction operation refers to all the effects that it produces on the structure (displacements, reactions of the supports, forces and stress).

The PH3 module can handle four types of ETATS of the structure, in a given construction phase:

- the PROBABLE state (of deformation and solicitation) is the result of accumulating all the effects of the operations declared to be cumulative, from the start of the construction up to and including the phase in question,
- the EXTREMAL state contains the minimum and maximum values taken by the probable state during the construction, and reduced to certain effects (reactions of supports, forces or stress) and certain components,
- the PONDERE (or CARACTERISTIQUE) state is an envelope state deduced from the probable state by individually weighting the favorable and unfavorable effects of the initial tensions of the cables, their losses of tension and the other permanent loadings,
- the PRECONTRAINT state is the probable state of the isostatic prestress solicitations in the beam elements.

FORFAITAIRE calculation

In this mode, the viscoelastic behaviors of the concrete and steels and their interactions are assumed to be PREDETERMINES by the reference regulations.

When the cables are tensioned, the PH3 module adds the PERTES ELASTIQUES deduced from the CONTRAINTES FINALES in the concrete, ESTIMEES and FOURNIES in the form of data at the start of the construction, to the instantaneous losses of tension.

The PERTES of tension in the cables due to the FLUAGE and RETRAIT of the concrete, and the RELAXATION of the prestressing steels are determined according to the SIMPLIFIEES formulas of the chosen regulation. Deferred losses are applied on demand, in one or several stages, according to the imposed RATIOS.

The REDISTRIBUTION of the forces by creep are evaluated by COMBINANT the current state, weighted by a coefficient $1 - v_2$, and the state of the structure obtained by applying the loads accumulated since the start of the construction, weighted by a coefficient v_2, with the current mechanical model. The resulting state becomes the current state.

RHEOLOGIQUE FIN calculation

In this mode, the viscoelastic behaviors of the concrete and the steels and their interactions are evaluated by taking account of the laws of evolution of the deformations over TEMPS.

The time variable is introduced in the form of a construction history that allows for its discretization (whether automatic or not), and the integration of the deferred phenomena. The distinction is made between:

- the period of CONSTRUCTION during which the time increments are 4 to 8 days,
- the period of VIEILLISSEMENT during which the time increments increase according to geometric progression.

The instantaneous losses of tension in the cables are deducted when they are tensioned. At the end of each time increment, the following are calculated and considered:

- the structural effects of the FLUAGE and RETRAIT of the concrete and the RELAXATION of the prestressing steels,
- the VARIATIONS in tension in the cables caused by the deformations of the concrete, which are applied to the concrete.

The REDISTRIBUTION of the forces is automatic and parallel to the chronological execution of the construction process.

During the definition of the construction history thanks to the command DATE, the user can enter the construction phases in a non-chronological order. During the execution of the module, PCP takes these phases into account in a chronological order if the option “Phases sorting” is active (Options/Phases sorting/Active). In this case, the software creates, in the case directory, a file *.PHASES_TRIEES.dec (where * is the name of the data file given by the user) including the construction phases in a chronological order. The user can check the construction with it.

LINEAIRE and non-LINEAIRE calculation

CALCUL LINEAIRE: By default, the structures are assumed to behave geometrically in small displacements and small rotations, i.e., with no second-order effects. In this context, certain entities may have a global ELASTOPLASTIQUE behavior: APPUIS, ARTICULATIONS and ELEMENTS. The BIARTICULES elements may have an intrinsic CHAINETTE behavior, but the displacements at the extremities are calculated in small displacements. The LINEAIRE and RHEOLOGIQUE FIN calculation modes are compatible.

CALCUL NON LINEAIRE: Geometric non-linearities are taken into consideration and the materials may have an ELASTOPLASTIQUE behavior. Like for the CALCUL LINEAIRE, certain entities may have a global ELASTOPLASTIQUE behavior: APPUIS, ARTICULATIONS and ELEMENTS. The BIARTICULES elements may have an intrinsic CHAINETTE behavior, but the displacements at the extremities are calculated in large displacements. The NON LINEAIRE and RHEOLOGIQUE FIN calculation modes are compatible.

The NON LINEAIRE calculation mode must be selected in order to make the FLAMBEMENT calculations (linear or otherwise).

The buckling modes are MODES type effects that can be produced by the ETU module or viewed by the RES module.

The following table summarizes the possibilities.

Function	**CALCUL LINEAIRE**	**CALCUL NON LINEAIRE**
Small displacements and small rotations	Yes	Yes
Large displacements and large rotations	No	Yes
APPUI/ARTICULATION ELASTOPLASTIQUE	Yes	Yes
ELEMENT ELASTOPLASTIQUE (global)	Yes	Yes
MATERIAU ELASTOPLASTIQUE	No	Yes
ACIER ELASTOPLASTIQUE	No	Yes
BIARTICULATION ELASTIQUE	Yes	Yes
BIARTICULATION ELASTOPLASTIQUE	Yes	Yes
BIARTICULATION CHAINETTE ELASTIQUE	Yes	Yes
BIARTICULATION CHAINETTE ELASTOPLASTIQUE	No	Yes
CALCUL RHEOLOGIQUE FIN	Yes	Yes
FLAMBEMENT LINEAIRE	No	Yes

Table 8.0 - Table of the linear and non-linear functions

General conditions

The use of the PH3 module is subject to the prior constitution of the general mechanical model by the PH1 module.

Data analysis mode

The phasing commands are analyzed in their totality and in the order of their introduction (interpretation). The PH3 module can be asked to simply verify them, without executing them. The verifications cover their syntax and their logical compatibility with the commands that have already been processed.

When implicit or explicit loadings are applied, the stability of the structure is only verified if the execution of the commands is requested.

With no end delimiter, each command in the PH3 module must begin on a new line and the first line break of the wording is to be observed. However, cutting data that follows a command header into lines, is unrestricted.

Editing

Commands are echoed as their interpretation proceeds; any erroneous commands are followed by error messages.

The results file contains the various effects and states that were implicitly requested to be created, plus certain results produced by the explicit editing commands (extreme states, tensions in the cables).

Contents

Contents (cont'd)

8.1 - PHASES

PHASES [DE CONSTRUCTION] [SUITE nom_structure]

Parameters

- nom_structure: name of a saved structure.

Functions

This command identifies a construction phasing file and starts a "session" using the PH3 module.

In the absence of the SUITE option, the PH3 module prepares to start a construction process, for which the operational options that are not selected by default must be established.

With the SUITE option, the PH3 module calls the saved structure in its state of activation and solicitation with the SAUVER nom_structure command, which was introduced in a previous session. The construction is then resumed at the moment when the backup process suspended it.

The acquired definitions and the operational options that applied when the backup was made are also restored, apart from any editing options (defined by the EDITER / NONEDITER commands), which are canceled and must be (re)introduced.

In these two cases, the VERIFIER command can be used.

Conditions of use

- ❐ Must be at the beginning of the command file.

Examples

```
PHASES  DE CONSTRUCTION
$ establishment of all the useful construction options
CALCULER  CONTRAINTES
............
EDITER  ....
$ start of the construction process
............
SAUVER  STRUCT_1
............
FIN
```

```
PHASES SUITE  STRUCT_1
$ restoration of the edit options only
EDITER  ....
$ resumption of the construction process (with the possibilty of saving
$ again)
............
FIN
```

Related commands

UNITES ; PARAMETRES RHEOLOGIQUES ; CAS DE CHARGE ; SAUVER ; FIN

8.2 - VERIFY

VERIFIER [nb_max_erreurs]

Parameters

- nb_max_erreurs: the maximum number of detectable errors, greater than or equal to 1. The control of the commands is interrupted as soon as this number of errors is reached. Its default value is 20.

Functions

As long as it does not come across a VERIFIER command, the PH3 module flags all the syntactical and logical errors that it detects and, if possible, performs all the requested analyses of the structure (EXECUTION mode).

This command causes an irreversible switch to the VERIFICATION mode. The following commands will be checked, but not executed.

The stability of the structure under the effect of the applied loads is only verified in EXECUTION mode.

Conditions of use

- This optional command can only be included once and can be introduced in any step of the construction process.

Methodological advice

- Always verify the commands before starting an important calculation, by inserting the VERIFIER command immediately after the PHASES command.
- Set nb_max_erreurs to 1 if you want to correct the errors one by one, thereby avoiding any errors that are caused by other errors.

Examples

```
PHASES  DE CONSTRUCTION
$ by default, module PH3 checks and executes the commands
$ (EXECUTION mode)
...........
FIN
```

```
PHASES  DE CONSTRUCTION
$ prefix this command with "$"
$ to inhibit its effects and return to EXECUTION mode
VERIFIER  1
...........
FIN
```

Related commands

UNITS

All commands that produce loadings, and/or results that are edited or recorded.

8.3 - UNITS

UNITES FORCES v_newtons $+\left\{\begin{matrix} \text{L\_FORCES} & \text{label\_forces} \\ \text{L\_CONTRAINTES} & \text{label\_contraintes} \end{matrix}\right\}$

Parameters

- v_newtons: value in Newtons of the unit used to express the forces, which will be used by the PH3, ENV, DYN and ETU modules.
- label_forces, label_contraintes: titles of the unit of forces and stress (chains of a maximum of 8 characters).

Functions

This command designates the unit used to express the forces. If it is absent, the kilo-deca-Newton is used by default (v_newtons = 10_000.0, label_forces = ‘kdaN’ and label_contraintes = ‘kdaN/m2’).

Conditions of use

- This optional command, which can only be used once, must immediately follow the PHASES command (or the VERIFIER command, if it is inserted in second position).
- This command is forbidden when the SUITE option of the PHASES command is used.

Examples

```
PHASES
$ the UNITES command is absent, by default the forces are expressed in
kdaN
...........
CHARGEMENT  IDENTIQUE  NOEUDS  50
1 A 50
0.0  0.0  -500.0      0.0  0.0  -1500.0
...........

PHASES
$ the forces are expressed in MN
UNITES  FORCES  1000000.0  L_FORCES  ’MN’  L_CONTRAINTES  ’Mpa’
...........
CHARGEMENT  IDENTIQUE  NOEUDS  50
1 A 50
0.0  0.0  -5.0      0.0  0.0  -15.0
...........
```

Related commands

PHASES ; VERIFIER ; MATERIAUX ELASTOPLASTIQUES

ACIERS ELASTOPLASTIQUES ; CHAINETTES ; AFFECTER CONTRAINTES

TENDRE ELEMENTS ; CHARGEMENT [IDENTIQUE] NOEUDS / EXTREMITES

CHARGEMENT POUTRE / [IDENTIQUE] ELEMENTS

8.4 – LINEAR CALCULATION

$$\text{CALCUL LINEAIRE}\ [\left\{\begin{array}{ll} \text{FORCES} & * \left\{\begin{array}{ll} \text{ITERATIONS} & \text{nb\_iter\_f} \\ \text{RELATIF} & \text{residu\_rela} \\ \text{MAXABSOLU} & \text{residu\_max} \\ \text{MINABSOLU} & \text{residu\_min} \end{array}\right\} \\ \text{ECHO\_TRACE} & \end{array}\right\}]$$

Parameters

- nb_iter_f: maximum number of iterations (200 by default).
- residu_rela: relative tolerance of the stop test for the convergence of the process that looks for the equilibrium of forces of the structure (1.0E-7 by default).
- residu_max: absolute residue in units of force required for the relative residue to be sufficient (1.0E-1 by default).
- residu_min: sufficient absolute residue in units of force, irrespective of the relative residue (1.0E-3 by default).

Functions

This command is used to modify the control parameters of a linear calculation.

It can also be used to adjust the values of the said parameters that have already been changed, or have default values, in the course of the calculation.

The ECHO_TRACE option displays the different steps of global equilibrium on the screen.

The relative and global values are calculated on the basis of the Euclidean standard of residual nodal vectors. The required value of the residue is between the absolute values fixed by this command. If the calculated relative value is below the required threshold, and the absolute value is below the residu_max, then the calculation converges. If the calculated relative value is greater than the required threshold, and the absolute value is below the residu_min, then the calculation converges. In this case, the convergence criterion is expressed as follows:

$$\left\{\begin{array}{ll} \mathrm{R}_{\mathrm{rel}} & \le \text{residu\_rela} \\ & \quad ET \\ \mathrm{R}_{\mathrm{abs}} & \le \text{residu\_max} \end{array}\right\} \quad OU \quad \mathrm{R}_{\mathrm{abs}} \le \text{residu\_min}$$

Conditions of use

- By default, the phasing module works in CALCUL LINEAIRE mode. Therefore, this command is only of any use to modify the default parameters.
- This optional command can be introduced several times, in various steps of the construction.
- Any following commands of this type can be used to modify one or more parameters attached to the non-linear calculation.

Example

```
PHASES
..............
$ modification of the echo convergence and display parameters
CALCUL  LINEAIRE ECHO FORCES ITER 100 RESI 1.e-5 MAXA 0.1 MINA 0.0001

..............
```

8.5 – NON-LINEAR CALCULATION

$$
\begin{array}{l}
\text{CALCUL} \\
\text{NONLINEAIRE}
\end{array}
\Bigg[\; {}^{*}\left\{
\begin{array}{l}
\text{SECTIONS} \; {}^{*}\left\{
\begin{array}{l}
\text{ITERATIONS nb\_iter\_s} \\
\text{RESIDU} \left\{ \begin{array}{l} \text{RELATIF residu\_relatif\_s} \\ \text{ABSOLU residu\_absolu\_s} \end{array} \right\} \\
\text{BRUTES} \\
\text{TRACE} \\
\text{METHODE} \left\{ \begin{array}{cc} \text{RAPIDE} & \\ \text{ROBUSTE} & \\ \text{TRES\_ROBUSTE} & \\ \text{FIXE} & \text{nu\_methode\_f} \\ \text{MULTIPLE} & [par\_mul] \end{array} \right\} \\
\text{CONTROLE ITERATION itc TEST tec FACTEUR fac} \\
\text{OSCILLATION ITERATION io TEST to MODULO mo} \\
\text{RUPTURE METHODE nru PIVOT pru FACTEUR fru} \\
\text{FACTEUR\_RELAXATION fre}
\end{array}
\right\} \\
\text{DEPLACEMENTS} \; {}^{*}\left\{ \begin{array}{l} \text{ITERATIONS nb\_iter\_d} \\ \text{TOLERANCE toler\_d} \end{array} \right\} \\
\text{FORCES} \; {}^{*}\left\{ \begin{array}{l} \text{ITERATIONS nb\_iter\_f} \\ \text{RESIDU residu\_f} \end{array} \right\} \\
\text{ECHO\_TRACE} \\
\text{PONDERATION\_ELEMENTS\_BIARTICULES pon\_b} \\
\text{D\_GEOMETRIQUE deg\_geo} \\
\text{D\_RIGIDITE deg\_rig} \\
\text{DIVISION\_CHARGE\_MAX nb\_fois} \\
\text{MODE no\_mode} \\
\text{[STRUCTURE nom\_structure] FACTEUR facteur}
\end{array}
\right\} \Bigg]
$$

$$
\text{[par\_mul]:Nu\_methode\_m MODULO mm} \left\{ \begin{array}{c} \text{CONTINUER} \\ \text{MEILLEURE} \\ \text{ZERO} \end{array} \right\} \text{QUADRATIQUE qu LINEAIRE li}
$$

Parameters

- nb_iter_s, toler_s: the maximum number of iterations and relative tolerance of the convergence stop test of the calculation of the deformations of the sections (100 and 1.0E-4 by default).
- residu_relatif_s : relative value of the equilibrium residual of the section with respect to the norm of the forces applied in the section (1.0E-6 by default)

- residu_absolu_s : absolute value of the equilibrium residual of the section expressed in the unit of computation force (See UNITE command). The default value is 0.0001 KdaN.
- nu_methode_f : Convergence method number set between 1 and 12 inclusive.
- nu_methode_m : Number of the first convergence method between 1 and 12 inclusive.
- qu et mm : qu value <1 such that if modulo (iteration, mm) = 0 and if the ratio of the averaged current residuals / previous averaged residues> qu, then PCP changes of quadratic method (by default, mm = 20 and that = 0.50)
- li et mm : li value <1 such that if modulo (iteration, mm) = 0 and if the ratio of current averaged residuals / averaged residuals> li, then PCP changes linear method (default, mm = 20 and li = 0.95).
- itc, tec et fac : if iteration> itc and value of the ratio current residue / previous residue> tec, the current strain is multiplied by the over-relaxation factor fac (default, itc = 15, tec = 1.5 and fac = 4/5).
- io, mo et fo : if iteration> io and modulo (iteration, mo) = 0 and || current strain + previous strain || <|| previous strain || / fo, the current strain is divided by the over-relaxation factor two (by default, io = 10, mo = 5 and fo = 3).
- nru : Number of method to use in case of break of cross-section (Default: nru = 10).
- pru : Minimum pivot value for cross-section break detection (Default: 1.e-12).
- fru : Sur-relaxation factor (<= 1) of the solution in case of failure (Default: Fru = 1).
- fre : Sur-relaxation factor (<= 1) of the solution before failure (Default: Fru = 1).
- nb_iter_d, toler_d: the maximum number of iterations and relative tolerance of the convergence stop test of the calculation of the displacements of the nodes (20 and 1.0E-5 by default).
- nb_iter_f, residu_f: maximum number of iterations and relative tolerance of the stop test for the convergence of the process that looks for the equilibrium of forces of the structure (1.0E-5 by default).
- pon_b: factor of numerical disruption between 0 and 1 of the transverse terms of the diagonal of the matrix of rigidity of the biarticulated elements and, therefore, of the chains to make it invertible or to numerically stabilize the transverse displacements (1.0E-6 by default).
- deg_geo, deg_rig: order of truncation of the displacements in the formulas that determine the equilibrium in the nodes, and of calculation of the matrix of tangential rigidity. These are exponents (that can equal 0, 3, 4 or 5), beyond which the terms expressed as displacements are ignored. A zero value indicates that no term of displacement is taken into consideration (calculation of the first order). deg_geo doit must be greater than or equal to deg_rig. If only deg_geo is provided, deg_rig is assumed to equal it. If only deg_rig is provided, the program adopts an exact geometric formulation. In the absence of these parameters, the exact formulations, without truncation, are adopted.
- nb_fois: the maximum number of times that the applied load must be divided by 2 when searching for the ultimate load (10 by default).
- no_mode, nom_structure, facteur: record number of the unitary buckling mode and name of the possible corresponding saved structure, to be taken into consideration as the initial geometric default of the structure, after having multiplied it in amplitude by the factor coefficient.

The parameters nb_fois, no_mode, nom_structure and facteur are used by the following ANALYSE CRITIQUE commands.

Functions

This command is used to start a non-linear calculation, possibly using certain parameters to control its execution and instabilities, or that will be useful in certain critical analyses in the future.

It can also be used to adjust the values of the said parameters that have already been changed, or have default values, in the course of the calculation.

With the SECTIONS BRUTES options, the calculation is made in raw sections of passive steels (as per BPEL). By default, the sections of passive steels are deducted from the sections of concrete. The sections of the cable sheathes are systematically deducted.

The TRACE option displays the different steps of the equilibrium of the sections on the screen. The ECHO_TRACE option displays the different steps of global equilibrium on the screen.

The PONDERATION value does not influence the equilibrium criteria of the nodes but, when increased in the convergence phase, it stabilizes the transverse displacements of only the nodes in equilibrium in the second order, such as the nodes of load-bearing cables of suspension bridges. On the other hand, this value influences the tangential rigidity used for a dynamic calculation or under operating loads. Therefore, once convergence has been reached for a given load case, the default value must be restored.

To find the balance of the sections, several convergence methods are available from the fastest to the most robust but the slowest:

- 1. Method of false position,
- 2. Digital tangent method,
- 3. Diagonal tangent method,
- 4. Mixed rope method 1,
- 5. Mixed rope method 2,
- 6. Strict rope method,
- 7. Broyden method,
- 8. DFP method,
- 9. BFGS method,
- 10. diagonal intersecting method,
- 11. Elastic method with acceleration,
- 12. Strict elastic method.

The first six methods have a so-called quadratic convergence speed and the last six have a linear or quasi-quadratic speed. The relaxation factor can reduce these speeds if it is set by the user to a value less than 1.

The MULTIPLE method starts with the method nu_methode_m which is incremented by 1 if the progression measured every mm iteration is less than qq or li depending on the type of method convergence speed. The maximum number of iterations can be increased by the user to allow PCP to test new methods. In the case of a method change, the initial deformation used is either the previous best (MEILLEURE), reset to ZERO or CONTINUE (default value) compared to the previous steps depending on the user's choices.

The RAPIDE method is actually a MULTIPLE method with the values of the corresponding option set to the default values (nb_iters = 200, mm = 20 qu = 0.50 and li = 0.95) and a starting method fixed to the false method position.

The ROBUSTE method is a MULTIPLE method with a starting method fixed to the false position method, nb_iters = 500, mm = 20 qu = 0.50 and li = 0.95.

The method TRES_ROBUSTE is a MULTIPLE method with a starting method fixed with the method of the false position, nb_iters = 1000, mm = 20 qu = 0.70 and li = 0.98.

The FIXE method is a MULTIPLE method with a starting method fixed to the chosen method, nb_iters = 1000, mm = 1000000 so the method is inchanged.

The additional parameters make it possible to manage the reduction of the residue (itc, tec and fac) or the strains oscillations (io, to and mo).

In case of section break, PCP switches to a strict string method or the method nru chosen by the user. The detection of the rupture is controlled by the pivot: pru. The over-relaxation factor makes it possible to further reduce the computation step so as to approach the breaking load as closely as possible.

Conditions of use

- This optional command can be introduced several times, in various steps of the construction.
- The first command of this type, which must appear before the start of the construction, causes an irreversible switch from the CALCUL LINEAIRE mode (selected by default) to the CALCUL NON LINEAIRE mode.
- Any following commands of this type can be used to modify one or more parameters attached to the non-linear calculation.

Methodological advice

- In the event of a discrepancy or a break, the ECHO and TRACE options must be activated to identify the elements concerned. PCP then displays the values of the residuals and convergence which then make it possible to modify the parameters of this command. These parameters can be modified locally during phasing and will remain active until the end of the calculation unless further modification.
- Changing residu_xx and toler_d affects the accuracy of calculations. Smaller values make calculations more accurate and values bigger, less accurate. Their modification is sometimes necessary but must always be minimized.
- The other parameters affect only the speed of convergence and not the final result. We will always begin by modifying these parameters before modifying the residuals and tolerances.

- Weighting factor of the biarticulated elements (and, therefore, the chains): as a general rule, the dichotomy on the loads applied in PCP in a non-linear calculation is coupled with an automatic increase of this coefficient. Therefore, modifying this coefficient is only useful to speed up the calculation process by fixing a minimal value, at the very start, to achieve convergence. In this case, remember to restore the initial value to 1.e-06 before saving the STRUCTURE, so that the calculations of the operating loads or the linear dynamic calculations use a non-disrupted tangential rigidity matrix.

Examples

```
PHASES
$ irreversible switch to CALCUL NON LINEAIRE mode
$ all the pre-initialized parameters return to their values by default
CALCUL  NONLINEAIRE
..............
..............
$ modification of the nb_iter_f and toler_s parameters
CALCUL  NONLINEAIRE  FORCES ITERATIONS 100  SECTIONS TOLERANCE 1.0E-6
..............

..............
..............
$ modification of the weighting factor
CALCUL  NONLINEAIRE  PONDERATION 0.001
..............
```

Related commands

MATERIAUX ELASTOPLASTIQUES ; ACIERS ELASTOPLASTIQUES

DEFINIR STRUCTURE POUSSEE ; SUSPENDRE / CONTINUER CONCORDANCE

SUSPENDRE / CONTINUER ANALYSE ; EDITER / NONEDITER

AFFECTER CONTRAINTES ; PLACER APPUIS ; REMPLACER APPUIS

ANALYSE CRITIQUE

8.6 – RHEOLOGICAL PARAMETERS

$$\text{PARAMETRES}\quad\text{RHEOLOGIQUES}\quad * \left\{ \begin{array}{l} \text{TEMPERATUR E} \quad \theta \\ \text{HYGROMETRI E} \quad \text{p\_hyg} \\ \text{C\_FLUAGE} \quad \text{c\_flu} \\ \text{C\_RETRAIT} \quad \text{c\_ret} \\ \text{C\_CISAILLE MENT} \quad \text{c\_cis} \\ \text{METHODE} \left\{ \begin{array}{l} \underline{\text{SUPERPOSIT ION}} \\ \text{KELVIN} \end{array} \right\} \end{array} \right\}$$

Parameters

- θ : ambient temperature (20.0 by default).
- p_hyg: hygrometric percentage of the air (70.0 by default).
- c_flu, c_ret, c_cis: weighting coefficients of the creep and shrinkage deformations and of shear force and torsion (1.0, 1.0 and 0.0 by default).

Functions

This command is used to define a group of parameters that can be used in CALCUL RHEOLOGIQUE FIN mode.

The METHODE option indicates the calculation method of the creeping.

- The method by default is the SUPERPOSITION method which is the Boltzman superposition method: if the loading $\sigma(t)$ history consists in n increments of loading $\Delta\sigma$, the creeping deformation from the loading time t_c is:

  $$\varepsilon^{fl}(t) = \sum_{i=0}^{n} J(t, t_{ci}) \Delta\sigma_i$$

  The function $J(t,t_{ci})$ is an increasing function in $(t-t_{ci})$ and null when $(t-t_{ci})$ is negative.

- The KELVIN method uses a Kelvin model, the principle of which is that all non-ageing viscous-elastic body can be modelised by grouping Kelvin chains in series. The creeping function can take this form:

  $$J(t, t_c) = \sum_{s=1}^{r} J_s \left(1 - \exp\left(-\frac{t - t_c}{\tau_s} \right) \right)$$

  Where τ_s and J_s are the time delay and the flexibility of each Kelvin chain s respectively (see document R7.01.01 of Code_aster).

Conditions of use

- ❐ This optional command can be introduced several times, but must always appear before the first DATE command. Therefore, it is forbidden to redefine rheological parameters in the course of the construction.
- ❐ This command is forbidden when the SUITE option of the PHASES command is used.

Examples

```
PHASES
...........
$ the shear deformations are not taken into consideration by default
$ in the adaptation phenomena; all the other parameters are fixed
$ by default, in particular the temperature (20 degrees Celsius)
PARAMETRES  RHEOLOGIQUES  HYGROMETRIE 80.0
...........
$ start of the construction
DATE  0
...........
```

Related commands

PHASES ; DATE

8.7 – ELASTOPLASTIC SUPPORT

APPUI ELASTOPLASTIQUE PARFAIT

$$\begin{Bmatrix}\text{TANGENT}\\ \text{CORDE}\\ \text{SECANT}\end{Bmatrix}\ \text{nom\_appui}\quad \text{COMPOSANTE}\ \ \text{no\_comp}\quad \text{PARAMETRES nb\_parametres}$$

$$\langle v\rangle_{\text{nb\_parametres}}$$

$$v = \mathrm{f}_0 \quad \mathrm{u}_1 \quad \mathrm{f}_1 \quad \mathrm{u}_2 \quad \mathrm{f}_2 \quad \dots \quad \mathrm{u}_\mathrm{n} \quad \mathrm{f}_\mathrm{n}$$

$$\text{attributs\_corde} = \begin{Bmatrix}\text{FMIN } \mathrm{f}_{\min}\\ \text{STOCHASTIQUE } \mathrm{v}_{\text{semence}}\\ \text{MIXTE } \mathrm{v}_{\text{corde}}\ \ \mathrm{v}_{\text{secant}}\\ \text{MI\_STOCHASTIQUE } \mathrm{v}_{\text{mix}}\end{Bmatrix}(\text{ITERATION\_DECLIC}\ \ i_{dec})$$

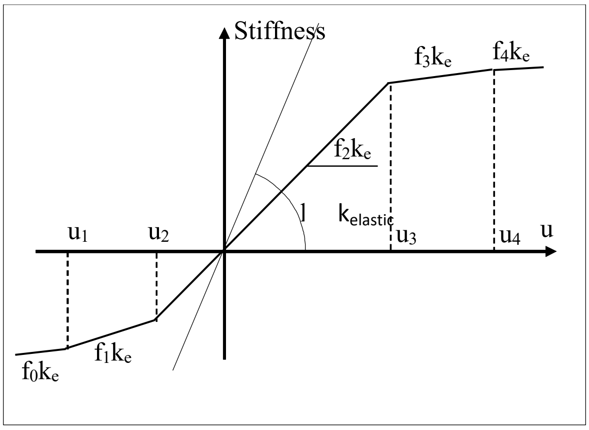


Figure 8.1 - Elastoplastic support

Parameters

- nom_appui: name of the support declared as elastoplastic.
- no_comp: number of the corresponding component of displacement (1 Dex, 2 Dey, 3 Dez, 4 Rox, 5 Roy, 6 Roz).
- nb_parametres: number of parameters defining the elastoplastic support.
- f_0: factor of elastic rigidity for a local displacement of the support node. Less than or equal to u_1.
- f_1: factor of elastic rigidity for a local displacement of the support node. Less than or equal to u_2 and greater than u_1.
- f_n: factor of elastic rigidity for a local displacement of the support node greater than u_n.

- f_{min}: minimum factor of rigidity between 0 and 1, used to build the chord rigidity from the elastic rigidity. If f_{min} equals 0, the convergence operator is determined by the option: CORDE. If it is greater than 0, PCP takes the maximum of the specified value and of the value obtained by applying f_{min}. By default, f_{min} is equal to 0.

$$f_{effectif\_rigidité} = \max\left(f_{corde}, f_{\min}\right)$$

- $v_{semence}$: real value of the seed for the stochastic calculus of the minimum factor of rigidity f_{min} between 0 and 1 used to build the chord rigidity from the elastic rigidity.

$$f_{\min} = f_{stochastique}(v_{semence})$$

$$f_{effectif\_rigidité} = \max\left(f_{corde}, f_{\min}\right)$$

- v_{corde} : value of the coefficient applied to the chord rigidity between 0 and 1. By default, v_{corde} is equal to 1.
- v_{secant} : value of the coefficient applied to the secant rigidity between 0 and 1. By default, v_{secant} is equal to 0.

$$f_{effectif\_rigidité} = v_{corde} f_{corde} + v_{sécant} f_{sécant}$$

- v_{mix} : positive or null real value of the seed for the stochastic calculus of v_{corde} and v_{secant}.

$$v_{corde} = f_{stochastique}(v_{mix})$$

$$v_{sécant} = 1 - v_{corde}$$

$$f_{effectif\_rigidité} = v_{corde} f_{corde} + v_{sécant} f_{sécant}$$

- i_{dec} : iteration number from which calculation options of the chord rigidity matrix are used. By default, i_{dec} is equal to 0.

Functions

This command is used to define a multi-linear elastoplastic support representing a unilateral support, a stop, a support with friction, an elastoplastic soil, a prestressed bush, etc. The stiffness of the support is specified for a component and in the form of a factor of elastic rigidity of the support corresponding to the general model. For a given interval of displacement, the final rigidity is the product of this factor by the elastic rigidity. The keywords TANGENT/CORDE/SECANT are used to specify the calculation mode of the convergence operator. These terms only influence the speed of the convergence, and not the final result. The CORDE option can be refined with the additional parameters. The option MIXTE or MI_STOCHASTIQUE allow the construction of the CORDE rigidity as a combination of the rigidity CORDE and SECANTE. The option FMIN or STOCHASTIQUE allow the reduction of the range of the convergence elastoplastic rigidity to a minimum value which is fixed or stochastic.

The elastoplastic aspect is taken into consideration when the specified support is installed. The displacement taken into consideration in the behavior is the current displacement of the node, determined after the installation of the support and projected in the local reference frame of the support by taking account of the eccentricity and orientation of the latter. The initial displacements of the nodes before the supports are installed do not intervene in the calculation of the reaction.

Conditions of use

- In the absence of this command, the supports have the rigidity initially defined by the APPUI commands of the general mechanical model.
- This command can be introduced in CALCUL LINEAIRE or CALCUL NON LINEAIRE mode, before any construction commands.
- Several components of the same support can successively be declared as elastoplastic.
- A component that is not declared as ELASTOPLASTIQUE remains elastic.
- It is forbidden to redefine the elastoplastic characteristics of a support for a given component.
- ELASTOPLASTIQUES supports keep this characteristic in the dynamic calculation model.

Methodological advice

- Initially, the support is generally declared as TANGENT which is the fastest option in the Newton numerical scheme. If the algorithm does not converge, the user can successively switch to CORDE mode by modifying the parameters of the calculation of the rigidity, then SECANT mode, if needed, to assure the convergence of the numerical scheme. The divergence often appears because of an infinite cyclic numerical oscillation. To avoid it, the user can numerically limit the chord rigidity with a minimum value f_{min} or by considering a combination of secant rigidity and chord rigidity. If the oscillation steel occurs, the user can stochastically determine f_{min} or the values of the combination.

Examples

```
$ unilateral support according to the component z (blocked according to
$ negative z)
f0=1 $ blocking of negative displacements
u1=0 $ blocking threshold
f1=0 $ unblocking of positive displacements
fmin=0
APPUI  ELASTOPLASTIQUE PARFAIT TANGENT APPUNZ COMPOSANTE 3 4
   f0 u1 f1 fmin
..............
..............
PLACER APPUI 1 $ Installation of the support
APPUNZ 10 6*0


$ Same unilateral support with chord rigidity
APPUI  ELAS PARFAIT CORDE APPUNZ COMP 3 PARA 3
f0 u1 f1


$ Same unilateral support with chord rigidity combined at the 25th
$iteration
fc=0.25 $ Chord rigidity factor
fs=0.75 $ Secant rigidity factor
APPUI  ELAS PARFAIT CORDE MIXTE fc fs ITER 25 APPUNZ COMP 3 PARA 3
f0 u1 f1

$ Same unilateral support with chord rigidity stochastically combined at
$the 25th iteration
fsemence=2
fs=0.75 $ Secant rigidity factor
APPUI  ELAS PARFAIT CORDE MI_ST fsemence ITER 25 APPUNZ COMP 3 PARA 3
```

```
f0 u1 f1

$ Plastic pile type ground support according to components x and y
f0=0     $ plastic ground
u1=0.01  $ plastification threshold
f1=1     $ elastic ground
ux=1 uy=2$ affected components
fmin=0
APPUI  ELASTOPLASTIQUE PARFAIT TANGENT Sol COMPOSANTE ux 6
   f0 (-u1) f1 u1 f0 fmin
APPUI  ELASTOPLASTIQUE PARFAIT TANGENT Sol COMPOSANTE uy 6
   f0 (-u1) f1 u1 f0 fmin
..............
..............
PLACER APPUI napp $ Installation of the ground
<< Sol (i) 6*0 > i = 1 a napp>
```

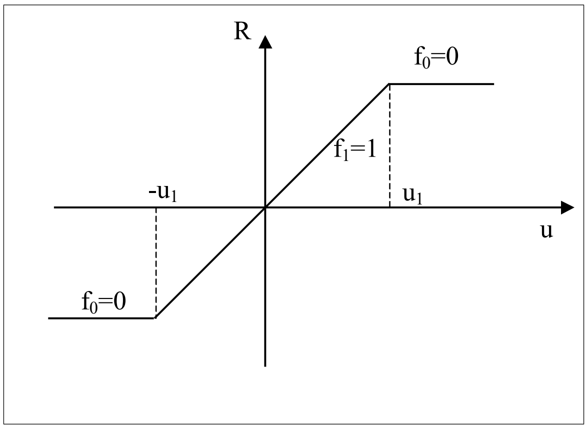


Figure 8.2 - Elastoplastic soil

```
$ Prestressed damper type support: F = Fp +kr u + c v
kr=10           $ stiffness of the spring defined in the model
fr=1            $ spring stiffness factor
fb=1000         $ housing stiffness factor
fbr= (fr+fb)    $ housing + spring stiffness factor
Fp= 100         $ prestress force
Up= (Fp/fbr/kr) $ prestress threshold in displacement
Ux=1            $ component X of the support
fmin=0
APPUI  ELASTOPLASTIQUE PARFAIT TANGENT AMORT COMPOSANTE ux 6
   fr (-up) fbr up fr fmin

$The dynamic module completes the definition of the damper with an
$AMORTISSEMENT DIRECT.
```

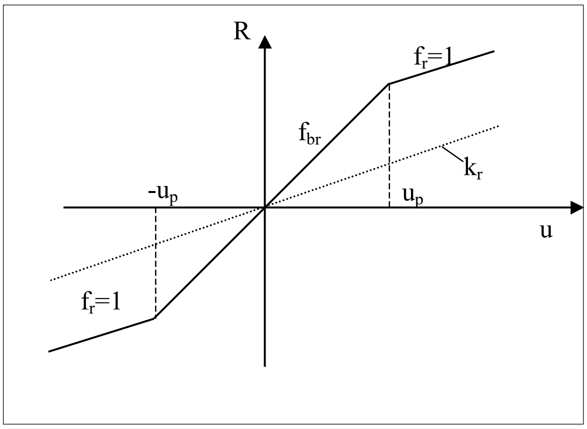


Figure 8.3 - Prestressed bush

8.8 – ELASTOPLASTIC ARTICULATION

$$\text{ARTICULATION} \quad \text{ELASTOPLASTIQUE} \left\{ \begin{array}{l} \text{PARFAITE} \\ \text{CYCLIQUE} \\ \text{TAKEDA} \end{array} \right\}$$

$$\left\{ \begin{array}{l} \text{TANGENTE} \\ \text{CORDE (attributs\_corde)} \\ \text{SECANT} \end{array} \right\} \text{nom\_articulation} \quad \text{COMPOSANTE} \;\; \text{no\_comp}$$

$$\text{PARAMETRES nb\_parametres}$$

$$\langle v \rangle_{\text{nb\_parametres}} \quad [\text{attributs\_takeda}]$$

$$v = \mathrm{f}_0 \quad \mathrm{u}_1 \quad \mathrm{f}_1 \quad \mathrm{u}_2 \quad \mathrm{f}_2 \quad \ldots \quad \mathrm{u}_n \quad \mathrm{f}_n$$

$$\text{attributs\_corde} = \left\{ \begin{array}{l} \text{FMIN } \mathrm{f}_{\text{min}} \\ \text{STOCHASTIQUE } \mathrm{v}_{\text{semence}} \\ \text{MIXTE } \mathrm{v}_{\text{corde}} \;\; \mathrm{v}_{\text{secant}} \\ \text{MI\_STOCHASTIQUE } \mathrm{v}_{\text{mix}} \end{array} \right\} (\text{ITERATION\_DECLIC } i_{dec})$$

$$\begin{array}{l} \text{attributs\_takeda} = \text{ALPHA } \mathrm{v}_{\text{alpha\_n}} \;\; \mathrm{v}_{\text{alpha\_p}} \;\; \text{BETA } \mathrm{v}_{\text{beta\_n}} \;\; \mathrm{v}_{\text{beta\_p}} \;\; \text{GAMMA } \mathrm{v}_{\text{gamma\_n}} \;\; \mathrm{v}_{\text{gamma\_p}} \\ \qquad\qquad\qquad (\text{EPSILON eps}) \quad (\text{NOMBRE\_ITERATIONS niter}) \end{array}$$

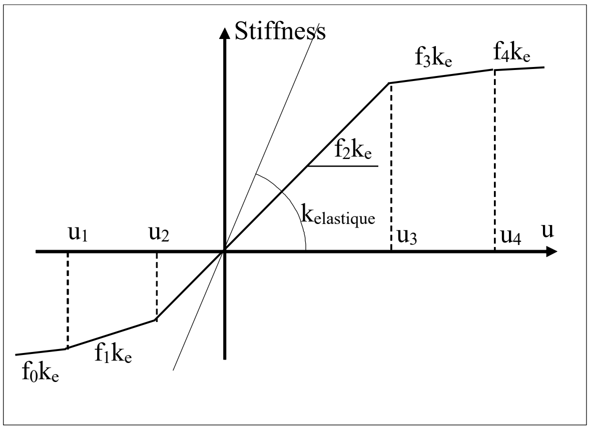


Figure 8.4 - Elastoplastic articulation

Parameters

- nom_articulation: name of the articulation declared as elastoplastic.
- no_comp: number of the corresponding component of displacement (1 Dex, 2 Dey, 3 Dez, 4 Rox, 5 Roy, 6 Roz).
- nb_parametres: number of parameters defining the elastoplastic articulation.
- f_0: factor of elastic rigidity for a local displacement of the articulation. Less than or equal to u_1.
- f_1: factor of elastic rigidity for a local displacement of the articulation. Less than or equal to u_2 and greater than u_1.
- f_n: factor of elastic rigidity for a local displacement of the articulation greater than u_n.
- f_{min}: minimum factor of rigidity between 0 and 1, used to build the tangential, secant or chord rigidity from the elastic rigidity. If f_{min} equals 0, the convergence operator is determined by the option: TANGENT/CORDE/SECANT. If it is greater than 0, PCP takes the maximum of the specified value and of the value obtained by applying f_{min}. By default, f_{min} is equal to 0.

$$f_{effectif\_rigidité} = \max\left(f_{corde}, f_{\min}\right)$$

- $v_{semence}$: real value of the seed for the stochastic calculus of the minimum factor of rigidity f_{min} between 0 and 1 used to build the chord rigidity from the elastic rigidity.

$$f_{\min} = f_{stochastique}(v_{semence})$$
$$f_{effectif\_rigidité} = \max\left(f_{corde}, f_{\min}\right)$$

- v_{corde} : value of the coefficient applied to the chord rigidity between 0 and 1. By default, v_{corde} is equal to 1.
- v_{secant} : value of the coefficient applied to the secant rigidity between 0 and 1. By default, v_{secant} is equal to 0.

$$f_{effectif\_rigidité} = v_{corde} f_{corde} + v_{sécant} f_{sécant}$$

- v_{mix} : positive or null real value of the seed for the stochastic calculus of v_{corde} and v_{secant}.

$$v_{corde} = f_{stochastique}(v_{mix})$$
$$v_{sécant} = 1 - v_{corde}$$
$$f_{effectif\_rigidité} = v_{corde} f_{corde} + v_{sécant} f_{sécant}$$

- i_{dec} : iteration number from which calculation options of the chord rigidity matrix are used. By default, i_{dec} is equal to 0.
- v_{alpha_n} et v_{alpha_p} : negative and positive alpha parameter of the TAKEDA law.
- v_{beta_n} et v_{beta_p} : negative and positive beta parameter of the TAKEDA law.
- v_{gamma_n} et v_{gamma_p} : negative and positive gamma parameter of the TAKEDA law.
- eps : precision of the TAKEDA law. By default, eps is equal to 1.e-12.
- iter : iteration number of calculations of the TAKEDA law. By default, iter is equal to 100.

Functions

This command is used to define a multi-linear elastoplastic articulation representing a unilateral articulation, a stop, an articulation with friction, an elastoplastic soil, a prestressed bush, etc. The stiffness of the articulation is specified for a component and in the form of a factor of elastic rigidity of the articulation corresponding to the general model. For a given interval of displacement, the final rigidity is the product of this factor by the elastic rigidity. The keywords TANGENTE/CORDE/SECANTE are used to specify the calculation mode of the convergence operator. These terms only influence the speed of the convergence, and not the final result. The CORDE option can be refined with the additional parameters. The option MIXTE or MI_STOCHASTIQUE allow the construction of the CORDE rigidity as a combination of the rigidity CORDE and SECANTE. The option FMIN or STOCHASTIQUE allow the reduction of the range of the convergence elastoplastic rigidity to a minimum value which is fixed or stochastic.

The elastoplastic aspect is taken into consideration when the specified articulation is installed. The displacement taken into consideration in the behavior is the variation of current displacement between the two nodes of the articulation, determined after the installation of the articulation and projected in the local reference frame of the articulation by taking account of the orientation of the latter. The initial displacements of the nodes before the articulations are installed do not intervene in the calculation of the forces supported by the articulation.

For the cyclical models and TAKEDA model, the diagram of the cycles is automatically produced by PCP in Csv format.

Perfect elastoplasticity

With the PARFAITE option, the elastoplastic law presents unloading branches which coincide whith the loading branches.

Cyclical elastoplasticity

With the CYCLIQUE option, the elastoplastic law is a bilinear law in which the unloading branch and the loading branch are different. The number of parameters is equal to 7 : f_0, u_1, f_1, u_2, f_2, u_3, f_3. They define the behavior law during the first loading. The law must be monotonously increasing.

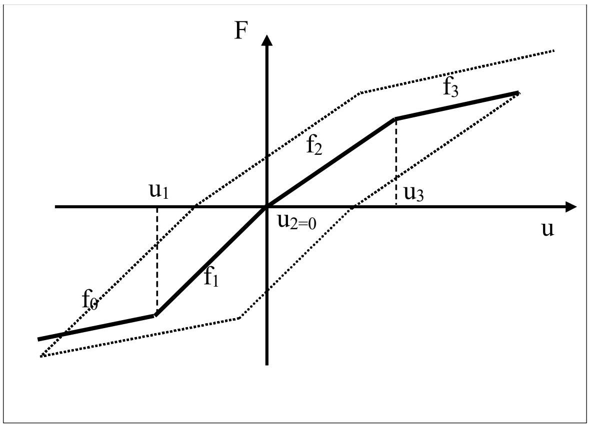


Figure 8.5 – Cyclic elastoplastic articulation

Where:

- f_0: negative plastic stiffness factor
- u_1: negative plasticity threshold
- f_1: negative elastic stiffness factor
- u_2: positive elasticity threshold (null in general)

- f_2: positive elastic stiffness factor
- u_3: positive plasticity threshold
- f_3: positive plastic stiffness factor

TAKEDA elastoplasticity

With the TAKEDA option, the elastoplastic law is a trilinear law of generalized TAKEDA type in which the unloading branch and the loading branch are different. Thus, the number of parameters is equal to 9 : f_0, u_1, f_1, u_2, f_2, u_3, f_3, u_4, f_4. The law must be monotonously and strictly increasing.

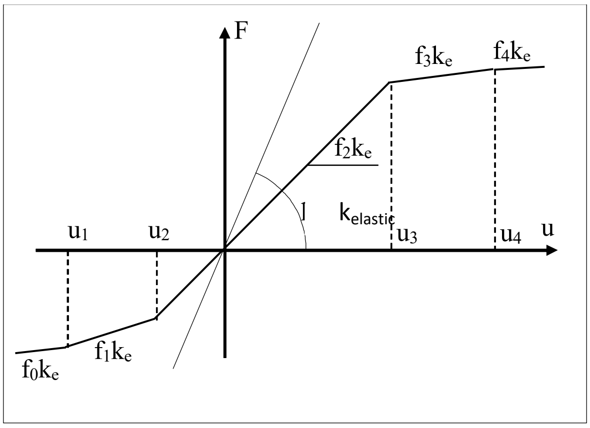


Figure 8.6 - TAKEDA type elastoplastic articulation

Where :

- f_0: negative plastic stiffness factor due to the steels alone (always>0)
- u_1: negative plasticity threshold of the steels
- f_1: negative plastic stiffness factor due to the plastified concrete and steels
- u_1: negative plasticity threshold of concrete
- f_2: positive elastic stiffness factor (1 in general)
- u_2: positive plasticity threshold of concrete
- f_3: positive plastic stiffness factor due to the plastified concrete and steels
- u_3: positive plasticity threshold of the steels
- f_4: positive plastic stiffness factor due to the steels alone (always >0)

The parameters alpha, beta and gamma allow to generalize the behavior considering the unloading module, the strength degradation and the pinch of the curves. The definition of these terms is as follow (cf. the IDARC calculation code by Y.J.PARK and al : Technical report N CEER-87-0008) :

- α^+ et α^- define the elastic unloading module, obtained by aiming a α^+R^+ or $-\alpha^-R^-$ ordinate point on the straight line passing through the origin with a slope equal to the initial rigidity.

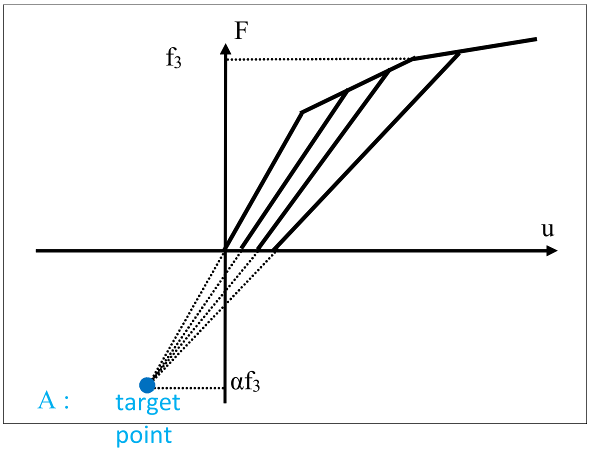


Figure 8.7 - Elastic unloading diagram

- β^+ et β^- define the strengh degradation under cyclic loading. During a unloading/loading cycle, the last target point is not the reached point of the previous cycle on the curve of first loading, but the one with an abscissa which is higher by a $\beta\Delta E/R$ factor where ΔE is the disspated energy during the previoust cycle.

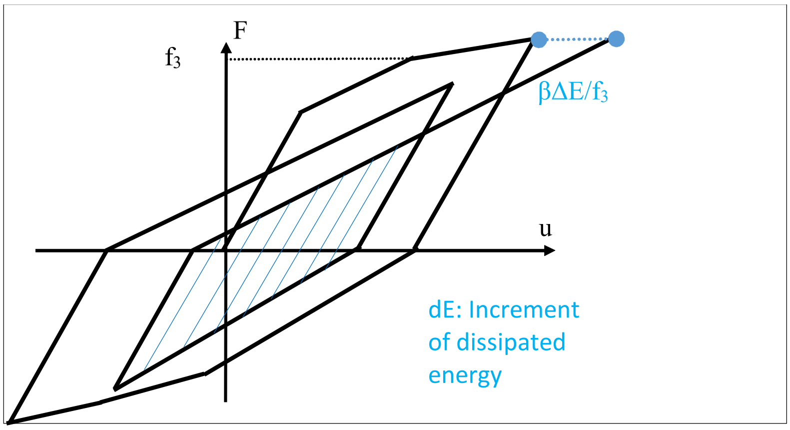


Figure 8.8 – Strengh degradation diagram

- γ^+ et γ^- define the pinch of the curves under a cyclic loading. When the sign of force changes, the γ^+R^+ or γ^-R^- ordinate point on the opposite unloading curve is aimed until the previous limit displacement is reached.

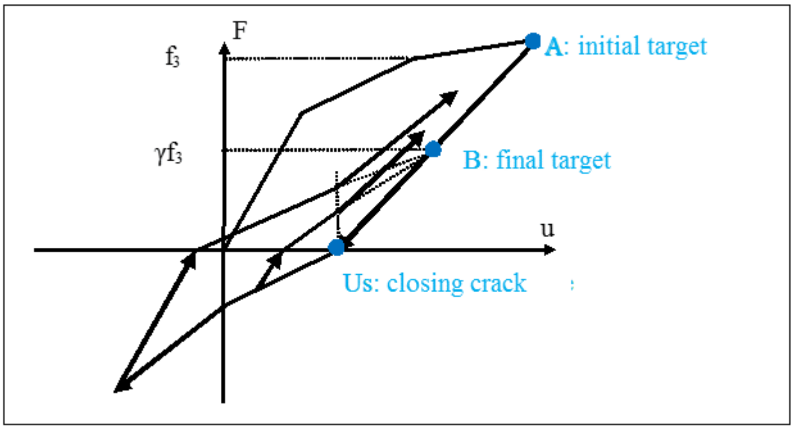


Figure 8.9 – Pinch diagram

TAKEDA elastoplasticity : examples

- Modified Takeda's model: α= 2, β= 0.1, γ= ∞
- Modified Clough's model: α= ∞, β= 0.1, γ= ∞
- Model oriented to the origin: α= 0, β= 0.1, γ= ∞
- Trilinear model of tee beam: α= 2, β= 0.1, γ= 0.5

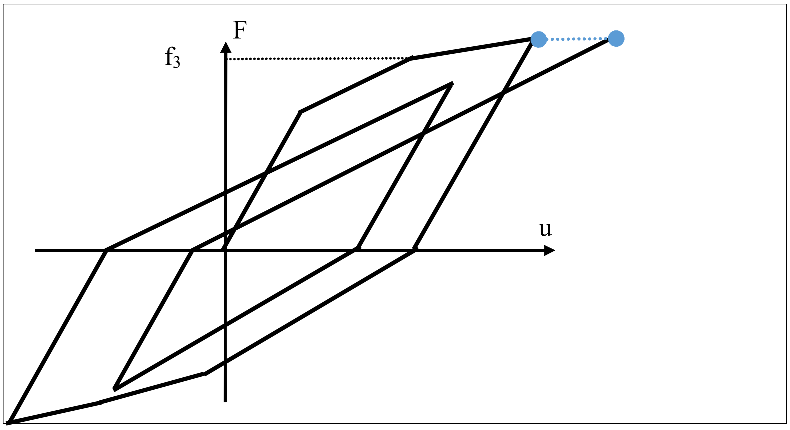


Figure 8.10 - Modified Takeda's model

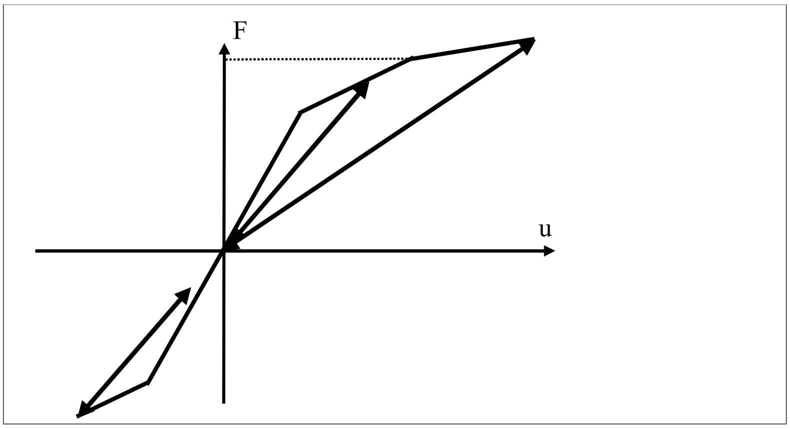


Figure 8.11 - Model oriented to the origin

Conditions of use

- In the absence of this command, the articulations have the rigidity initially defined by the ARTICULATION commands of the general mechanical model.
- This command can be introduced in CALCUL LINEAIRE or CALCUL NON LINEAIRE mode, before any construction commands.
- Several components of the same articulation can successively be declared as elastoplastic.
- A component that is not declared as ELASTOPLASTIQUE remains elastic.
- It is forbidden to redefine the elastoplastic characteristics of an articulation for a given component.
- ELASTOPLASTIQUES articulations keep this characteristic in the dynamic calculation model.

Methodological advice

- Initially, the support is generally declared as TANGENT which is the fastest option in the Newton numerical scheme. If the algorithm does not converge, the user can successively switch to CORDE mode by modifying the parameters of the calculation of the rigidity, then SECANT mode, if needed, to assure the convergence of the numerical scheme. The divergence often appears because of an infinite cyclic numerical oscillation. To avoid it, the user can numerically limit the chord rigidity with a minimum value f_{min} or by considering a combination of secant rigidity and chord rigidity. If the oscillation steel occurs, the user can stochastically determine f_{min} or the values of the combination.
- The cyclic Takeda laws can be independently used for each bending or shearing direction. Thus, they provide effectively the seismic behavior of a hinged sugjected to bending or shearing for any particular normal strength
- In the case of cyclical models, the user can check the behavior by visualize the corresponding Csv file.

Examples

```
$ Unilateral articulation as per the component z (blocked per negative z)
f0=1 $ blocking of negative displpacements
u1=0 $ blocking threshold
f1=0 $ unblocking of positive displacements
fmin=0
ARTICULATION  ELASTOPLASTIQUE PARFAIT TANGENT ARTUNZ COMPOSANTE 3 4
   f0 u1 f1 fmin
..............
PLACER ARTICULATION  1 $ installation of the articulation
ARTUNZ 10 3*0


$ Same unilateral articulation with chord rigidity
ARTI  ELAS PARFAIT CORDE ARTUNZ COMP 3 PARA 3
f0 u1 f1

$ Same unilateral articulation with chord rigidity combined at the 25th $
iteration
fc=0.25 $ Chord rigidity factor
fs=0.75 $ Secantrigidity factor
ARTI  ELAS PARFAIT CORDE MIXTE fc fs ITER 25 ARTUNZ COMP 3 PARA 3
f0 u1 f1
```

```
$ Plastic pile type articulation according to components x and y
f0=0     $ plastic ground
u1=0.01  $ plastification threshold
f1=1     $ elastic ground
ux=1 uy=2$affected components
fmin=0
ARTI  ELASTOPLASTIQUE PARFAIT TANGENT Sol COMPOSANTE ux 6
   f0 (-u1) f1 u1 f0 fmin
ARTI  ELASTOPLASTIQUE PARFAIT TANGENT Sol COMPOSANTE uy 6
   f0 (-u1) f1 u1 f0 fmin
..............
..............
PLACER ARTI napp $ installation of the ground
<< Sol (i) 3*0 > i = 1 a napp>
```

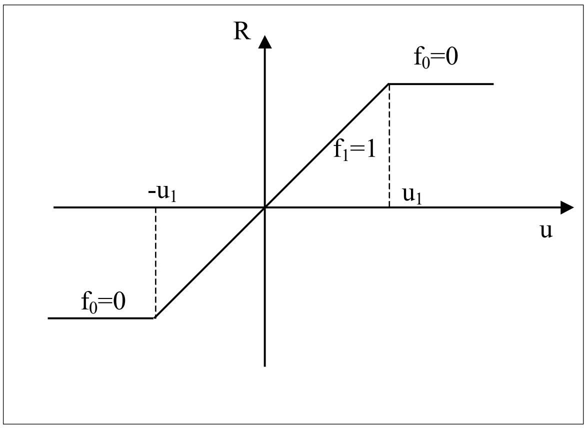


Figure 8.12 - Elastoplastic soil

```
$ Prestressed damper type articulation: F = Fp +kr u + c v
kr=10           $ stiffness of the spring defined in the model
fr=1            $ spring stiffness factor
fb=1000         $ housing stiffness factor
fbr= (fr+fb)    $ housing + spring stiffness factor
Fp= 100         $ prestress force
Up= (Fp/fbr/kr) $ prestress threshold in displacement
Ux=1            $ component X of the articulation
fmin=0
ARTI  ELASTOPLASTIQUE PARFAIT TANGENT AMORT COMPOSANTE ux 6
   fr (-up) fbr up fr fmin

$The dynamic module completes the definition of the damper with an
$AMORTISSEMENT DIRECT.
```

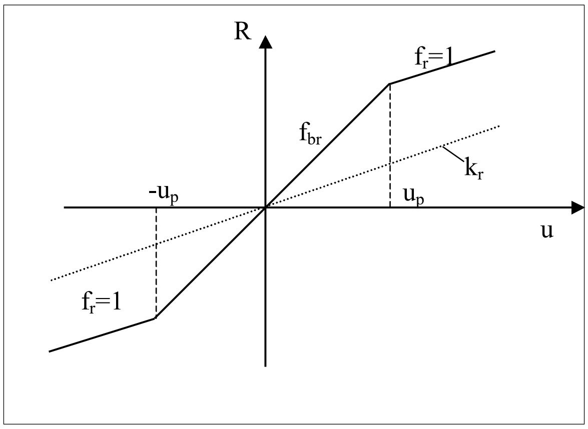


Figure 8.13 - Prestressed bush

```
$ Articulation representing a plastic bending hinge according to the
TAKEDA $ law modified by the rotational component y.

f0 = 0.001     $ negative plastic stiffness factor due to the steels
alone
u1 = -0.000002 $ negative plasticity threshold of the steels
f1 = 0.5       $ negative plastic stiffness factor due to the plastified
             $ concrete and steels
u1 = -0.000001 $ negative plasticity threshold of concrete
f2 = 1         $ elastic stiffness factor
u2 = 0.000001  $ positive plasticity threshold of concrete
f3 = 0.5       $ positive plastic stiffness factor due to the plastified
             $ concrete and steels
u3 = 0.000002  $ positive plasticity threshold of the steels
f4 = 0.001     $ positive plastic stiffness factor due to the steels
alone
             $ (always >0)

v_Alpha = 2
v_Beta  = 0.1
v_Gamma = 1000

ARTICULATION ELASTOPLASTIQUE TAKEDA TANGENT ROTY COMPOSANTE 5 PARAMETRES
9
f0 u1 f1 u2 f2 u3 f3 u4 f4
ALPHA v_alpha v_alpha BETA v_beta _v_beta GAMMA v_gamma v_gamma
..............
..............
PLACER ARTICULATION  1 $ Implementation of the articulation

ROTY 10 3*0
```

8.9 – ELASTOPLASTIC ELEMENT

$$\text{ELEMENT} \quad \text{ELASTOPLASTIQUE} \quad \left\{\begin{array}{l}\text{PARFAIT}\\ \text{CYCLIQUE}\\ \text{TAKEDA}\end{array}\right\}$$

$$\left\{\begin{array}{l}\text{TANGENT}\\ \text{CORDE (attributs\_corde)}\\ \text{SECANT}\end{array}\right\} \ \text{no\_element (SECTIONS } n_{sect}\text{)} \quad \text{COMPOSANTE} \ \ \text{no\_comp}$$

$$\text{PARAMETRES nb\_parametres}$$

$$\langle v \rangle_{\text{nb\_parametres}} \quad [\text{attributs\_takeda}]$$

$$v = \mathrm{f}_0 \quad \mathrm{u}_1 \quad \mathrm{f}_1 \quad \mathrm{u}_2 \quad \mathrm{f}_2 \quad \ldots \quad \mathrm{u}_\mathrm{n} \quad \mathrm{f}_\mathrm{n}$$

$$\text{attributs\_corde} = \left\{\begin{array}{l}\text{FMIN } \mathrm{f}_{\mathrm{min}}\\ \text{STOCHASTIQUE } \mathrm{v}_{\mathrm{semence}}\\ \text{MIXTE } \mathrm{v}_{\mathrm{corde}} \ \ \mathrm{v}_{\mathrm{secant}}\\ \text{MI\_STOCHASTIQUE } \mathrm{v}_{\mathrm{mix}}\end{array}\right\} (\text{ITERATION\_DECLIC } i_{dec})$$

$$\text{attributs\_takeda=ALPHA } \mathrm{v}_{\mathrm{alpha\_n}} \ \ \mathrm{v}_{\mathrm{alpha\_p}} \ \ \text{BETA } \mathrm{v}_{\mathrm{beta\_n}} \ \ \mathrm{v}_{\mathrm{beta\_p}} \ \ \text{GAMMA } \mathrm{v}_{\mathrm{gamma\_n}} \ \ \mathrm{v}_{\mathrm{gamma\_p}}$$

$$\text{(EPSILON eps) (NOMBRE\_ITERATIONS niter)}$$

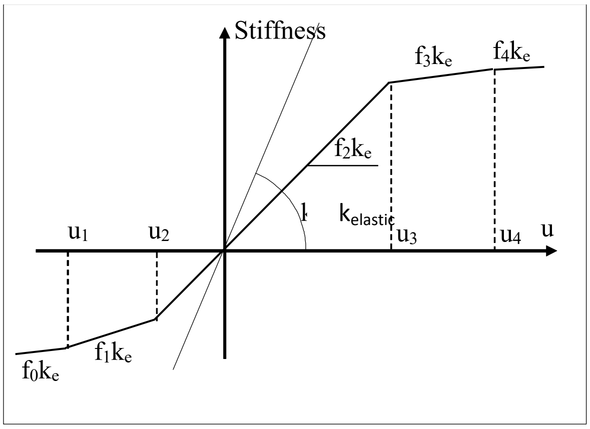


Figure 8.14 - Elastoplastic element

Parameters

- no_element: number of the element declared as elastoplastic.
- no_comp: number of the corresponding component of deformation (1 Dex, 2 Dey, 3 Dez, 4 Rox, 5 Roy, 6 Roz).
- nb_parametres: number of parameters defining the elastoplastic element.
- f_0: factor of elastic rigidity for a local deformation of the element. Less than or equal to u_1.
- f_1: factor of elastic rigidity for a local deformation of the element. Less than or equal to u_2 and greater than u_1.
- f_n: factor of elastic rigidity for a local deformation of the element greater than u_n.
- f_{min}: minimum factor of rigidity between 0 and 1, used to build the tangential, secant or chord rigidity from the elastic rigidity. If f_{min} equals 0, the convergence operator is determined by the option: TANGENT/CORDE/SECANT. If it is greater than 0, PCP takes the maximum of the specified value and of the value obtained by applying f_{min}. By default, f_{min} is equal to 0.

$$f_{effectif\_rigidité} = \max\left(f_{corde}, f_{\min}\right)$$

- n_{sec} : Number of points of longitudinal integration for the calculation of the rigidity of the element. This parameter is only used in a CALCUL NON LINEAIRE.
- $v_{semence}$: real value of the seed for the stochastic calculus of the minimum factor of rigidity f_{min} between 0 and 1 used to build the chord rigidity from the elastic rigidity.

$$f_{\min} = f_{stochastique}\left(v_{semence}\right)$$
$$f_{effectif\_rigidité} = \max\left(f_{corde}, f_{\min}\right)$$

- v_{corde} : value of the coefficient applied to the chord rigidity between 0 and 1. By default, v_{corde} is equal to 1.
- v_{secant} : value of the coefficient applied to the secant rigidity between 0 and 1. By default, v_{secant} is equal to 0.

$$f_{effectif\_rigidité} = v_{corde} f_{corde} + v_{sécant} f_{sécant}$$

- v_{mix} : positive or null real value of the seed for the stochastic calculus of v_{corde} and v_{secant}.

$$v_{corde} = f_{stochastique}\left(v_{mix}\right)$$
$$v_{sécant} = 1 - v_{corde}$$
$$f_{effectif\_rigidité} = v_{corde} f_{corde} + v_{sécant} f_{sécant}$$

- i_{dec} : iteration number from which calculation options of the chord rigidity matrix are used. By default, i_{dec} is equal to 0.
- v_{alpha_n} et v_{alpha_p} : negative and positive alpha parameter of the TAKEDA law.
- v_{beta_n} et v_{beta_p} : negative and positive beta parameter of the TAKEDA law.
- v_{gamma_n} et v_{gamma_p} : negative and positive gamma parameter of the TAKEDA law.
- eps : precision of the TAKEDA law.
- iter : iteration number of calculations of the TAKEDA law. By default, iter is equal to 100.

Functions

This command is used to define a multi-linear elastoplastic element. The stiffness of the element is specified for a component and in the form of a factor of elastic rigidity of the element corresponding to the general model. For a given interval of deformation, the final rigidity is the product of this factor by the elastic rigidity. The keywords TANGENTE/CORDE/SECANTE are used to specify the calculation mode of the convergence operator. These terms only influence the speed of the convergence, and not the final result. The CORDE option can be refined with the additional parameters. The option MIXTE or MI_STOCHASTIQUE allow the construction of the CORDE rigidity as a combination of the rigidity CORDE and SECANTE. The option FMIN or STOCHASTIQUE allow the reduction of the range of the convergence elastoplastic rigidity to a minimum value which is fixed or stochastic.

The influenced terms of rigidity are respectively Ax, Ay, Az, Ix, Iy and Iz for the deformations dex, dey, dez, rox, roy, roz, measured in meters per meter for displacements and in radians per meter for rotations.

The elastoplastic aspect is taken into consideration when the specified element is activated.

In a CALCUL LINEAIRE, all elements have the same law along their entire length. The mean deformation is evaluated in the local reference frame of the element.

In a CALCUL NON LINEAIRE, the deformation is evaluated at each point of integration in the INTRINSEQUE reference frame of the element. If this number is 1, the mean deformation of the element is used to calculate its rigidity.

For the cyclical models and TAKEDA model, the diagram of the cycles is automatically produced by PCP in Csv format.

Perfect elastoplasticity

With the PARFAITE option, the elastoplastic law presents unloading branches which coincide whith the loading branches.

Cyclical elastoplasticity

With the CYCLIQUE option, the elastoplastic law is a bilinear law in which the unloading branch and the loading branch are different. The number of parameters is equal to 7 : f_0, u_1, f_1, u_2, f_2, u_3, f_3. They define the behavior law during the first loading. The law must be monotonously increasing.

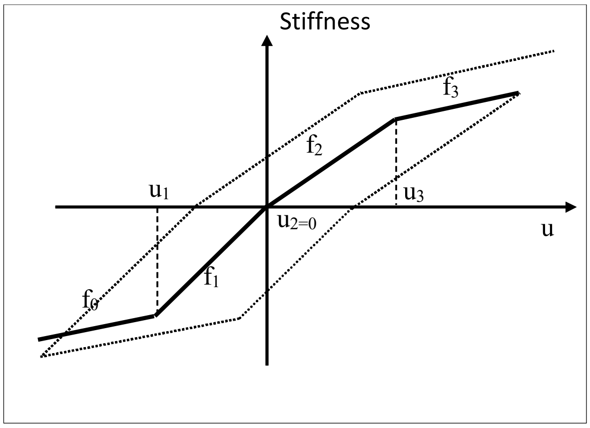


Figure 8.15 - Cyclic elastoplastic articulation

Where:

- f_0: negative plastic stiffness factor
- u_1: negative plasticity threshold
- f_1: negative elastic stiffness factor
- u_2: positive elasticity threshold (null in general)
- f_2: positive elastic stiffness factor
- u_3: positive plasticity threshold
- f_3: positive plastic stiffness factor

TAKEDA elastoplasticity

See the equivalent option of the command ARTICULATIONS ELASTOPLASTIQUE.

Conditions of use

- ❐ In the absence of this command, the elements have the rigidity initially defined in the general mechanical model.
- ❐ This command can be introduced in CALCUL LINEAIRE or CALCUL NON LINEAIRE mode, before any construction commands.
- ❐ Several components of the same element can successively be declared as elastoplastic.
- ❐ For BIARTICULE, elements, only the number 1 component is permitted.
- ❐ A component that is not declared as ELASTOPLASTIQUE remains elastic.
- ❐ It is forbidden to redefine the elastoplastic characteristics of an element for a given component.
- ❐ It is forbidden to declare an element as ELASTOPLASTIQUE if its material is declared as ELASTOPLASTIQUE, and vice versa.
- ❐ It is forbidden to declare an element as ELASTOPLASTIQUE if its material is declared as creeping or shrinking.
- ❐ It is forbidden to declare an element containing cables or steels as ELASTOPLASTIQUE.
- ❐ It is forbidden to declare a chain as ELASTOPLASTIQUE in CALCUL LINEAIRE mode. But this is possible in CALCUL NON LINEAIRE mode.
- ❐ ELASTOPLASTIQUES elements keep this characteristic in the dynamic calculation module.

Methodological advice

- ❐ Initially, the support is generally declared as TANGENT which is the fastest option in the Newton numerical scheme. If the algorithm does not converge, the user can successively switch to CORDE mode by modifying the parameters of the calculation of the rigidity, then SECANT mode, if needed, to assure the convergence of the numerical scheme. The divergence often appears because of an infinite cyclic numerical oscillation. To avoid it, the user can numerically limit the chord rigidity with a minimum value f_{min} or by considering a combination of secant rigidity and chord rigidity. If the oscillation steel occurs, the user can stochastically determine f_{min} or the values of the combination.

- In a CACUL NON LINEAIRE, the number of points of integration can be set at 5 for a conventional beam element. For an element likely to represent a plastic hinge subjected to an earthquake, its length must equal the width of the plastic hinge and the number of integration points is taken to equal 1, so that all of the element has non-linear behavior with the same characteristics.
- The options CYCLIQUE and TAKEDA can be used for seismic surveys to provide the concrete hysteretic behavior on the entire length of the corresponding plastic hinge.
- The options CYCLIQUE and TAKEDA can be independently used for each bending or shearing direction. Thus, they provide effectively the seismic behavior of a hinged sugjected to bending or shearing for any particular normal strength.
- In the case of cyclical models, the user can check the behavior by visualize the corresponding Csv file.

Example

```
$ Foot element of a pile that is elastoplastic in earthquakes
f0=0     $ plastic ball joint
u1=0.01  $ plastification threshold
f1=1     $ elastic ball joint
ux=1 uy=2 $ affected components
fmin=0
nbsec=5
ELEMENT ELASTOPLASTIQUE PARFAIT TANGENT ROTULE COMPOSANTE ux 6
   f0 (-u1) f1 u1 f0 fmin
ELEMENT ELASTOPLASTIQUE PARFAIT TANGENT ROTULE COMPOSANTE uy 6
   f0 (-u1) f1 u1 f0 fmin nbsec
..............
..............

$ Same element with a chord rigidity
ELEMENT ELAS PARFAIT CORDE ROTULE SECT nbsec COMPOSANTE uy PARAMETRES 5
f0 (-u1) f1 u1 f0

$ Same element with a chord rigidity combined at the 25th iteration
fc=0.25 $ Facteur de rigidite corde
fs=0.75 $ Facteur de rigidite sécante
ELEMENT ELAS PARF CORDE MIXT fc fs ITER 25 ROTU SECT nbsec COMP uy PARA 5
f0 (-u1) f1 u1 f0
```

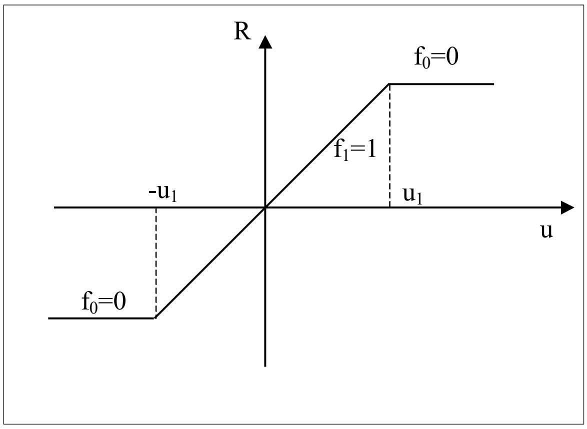


Figure 8. 16 - Elastoplastic element

```
$ Articulation representing a plastic bending hinge according to the
TAKEDA $ law modified by the rotational component y.

ROTULE=10       $ Hinge number
f0 = 0.001      $ negative plastic stiffness factor due to the steels
alone
u1 = -0.000002 $ negative plasticity threshold of the steels
f1 = 0.5        $ negative plastic stiffness factor due to the plastified
             $ concrete and steels
u1 = -0.000001 $ negative plasticity threshold of concrete
f2 = 1          $ positive elastic stiffness factor
u2 = 0.000001  $ positive plasticity threshold of concrete
f3 = 0.5        $ positive plastic stiffness factor due to the plastified
             $ concrete and steels
u3 = 0.000002  $ positive plasticity threshold of the steels
f4 = 0.001      $ positive plastic stiffness factor due to the steels
alone
             $ (always >0)

v_Alpha = 2
v_Beta  = 0.1
v_Gamma = 1000

ELEMENT ELASTO TAKEDA TANGENT ROTULE SECT nbsec COMPOSANTE 5 PARAMETRES 9
f0 u1 f1 u2 f2 u3 f3 u4 f4
ALPHA v_alpha v_alpha BETA v_beta _v_beta GAMMA v_gamma v_gamma
```

8.10 – ELASTOPLASTIC MATERIALS

MATERIAUX ELASTOPLASTIQUES part_materiaux

$$\text{TYPE}\left\{\begin{array}{lll} \left\{\begin{array}{l}\text{LINEAIRE}\\ \text{BILINEAIRE}\end{array}\right\} & \text{SOUS\_TYPE} & \left\{\begin{array}{l}\text{PARFAIT}\\ \text{FRAGILE}\end{array}\right\} \\ \left\{\begin{array}{l}\text{BPEL}\\ \text{BPHP}\\ \text{EC02}\end{array}\right\} & \text{SOUS\_TYPE} & \left\{\begin{array}{l}\text{SARGIN}\\ \text{PARABOLE}\end{array}\right\} \\ \text{EC08} & \text{SOUS\_TYPE} & \text{BETON} \\ \left\{\begin{array}{l}\text{PARABOLE}\\ \text{S\_BRANCHE}\\ \text{SARGIN}\end{array}\right\} & \text{SOUS\_TYPE} & \text{SIMPLE} \\ \text{ACIER} & \text{SOUS\_TYPE} & \left\{\begin{array}{l}\text{BILINEAIRE}\\ \text{BA83}\end{array}\right\} \end{array}\right\}$$

[EPS0 ε_0] [SIG0 σ_0] [EPSU ε_u] [SIGU σ_u]

[SIGT σ_t] [EPTU ε_{tu}] [SITU σ_{tu}]

[K1 k_1] [K2 k_2]

[ADEF af_ε] [ACON af_σ]

[SECTIONS nb_sections]

[PASY py] [PASZ pz] [NBPY nbpy] [NBPZ nbpz]

Parameters

- part_materiaux: selection particle (similar to a name). All the materials defined by the MATERIAU commands of the GE1 module (see Chapter 3) or PH1 module (see Chapter 6), and whose names contain this particle are affected.
- ε_0: deformation at the start of the plastic plateau of the diagram representing the behavior law of the selected materials, or stress peak (Figure 8.1).
- σ_0: stress at the start of the plastic plateau, or resistance to compression.
- ε_u: ultimate deformation, or at rupture in compression.
- σ_u: rupture stress in compression.
- σ_t: resistance to traction (in an absolute value).
- k_1: coefficient of Sargin's law.
- k_2: coefficient of Sargin's law.
- ε_{tu}: ultimate deformation, or at rupture in traction (in an absolute value).
- σ_{tu}: rupture stress in traction (in an absolute value).
- af_ε: depending on the case, the coefficient to be applied as an affinity factor, to the diagram of stress deformations in the axis of the deformations, or only as a denominator in the modulus of elasticity tangential to the origin (see Table 8.9 below). This value is usually greater than 1. For example, it may be the creep coefficient of the BPEL or the coefficient γ_{cE} of the Eurocode defined in EC2-1-1 5.8.6 (3).

- af_σ: coefficient of affinity- to be applied to the diagram of stress deformations, in the axis of the stress. This value is usually less than 1, because it corresponds to the partial security coefficient in resistance of the material.

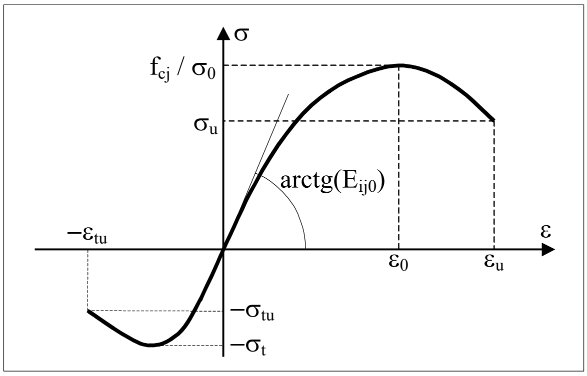


Figure 8. 17 - Base material. Diagram of typical stress deformations

- nb_sections: the number of calculation sections, regularly spaced, to be taken into consideration to integrate the material non-linearity along each of these elements, made up of the selected materials (origins and ends included). This value must be odd and at least equal to 3 (the default value).

The bending rigidity of the sections of the elements, which are selected implicitly by their constituent materials, are calculated by cutting them up into "initial" trapeziums according to their main axes of inertia, Gy and Gz, which are in turn subdivided into "elementary" trapeziums of the same height, that correspond to a discretization interval.

- py, pz, nbpy, nbpz: the effective interval of discretization (0.05 by default), and the number of elementary trapeziums to be placed in each initial trapezium, according to the Gy and Gz axes of the sections.

When ni nbpy and ni nbpz are not provided, each initial trapezium of height hy or hz, in the axis in question, is cut according to an odd number of intermediate lines belonging to the interval [3, 21], and the closest of hy/py or hz/pz. When only one of these values is provided, the other is 1. When at least one of these values is provided, the height of each elementary trapezium is chosen to be equal to min(py, hy/nbpy) or min(pz, hz/nbpz).

Certain parameters, such as f_{cj}, are deduced from the current Young's modulus. For non-aging modules, this equals the para_supp$_1$ module of the MATERIAU command. For an aging module, the current Young's modulus is deduced from para_supp$_1$, according to the current date and the date of casting.

The table below shows the parameters to be provided, according to the type and the subtype of the selected behavior law.

No.	Type	Subtype	Description	Parameters
01	LINEAR	PERFECT	perfect linear behavior law without rupture limit	None
02	LINEAR	FRAGILE	fragile linear behavior law with rupture limit	σ_0, σ_t
03	BILINEAR	PERFECT	perfect bilinear behavior law with plastic plateau and without rupture limit	σ_0, σ_t
04	BILINEAR	FRAGILE	fragile bilinear behavior law with plastic plateau	σ_0, ε_u, σ_u, σ_t, ε_{tu}, σ_{tu}
05	BPEL	SARGIN	Sargin-type law of the BPEL	ε_u [, af_ε] [, af_σ]
06	BPEL	PARABOLA	parabola-rectangle-type law of the BPEL	ε_u [, af_ε] [, af_σ]
07	BPHP	SARGIN	Sargin-type law of the BPEL, as per Annex 14 on HP concretes	ε_u [, af_ε] [, af_σ]
08	BPHP	PARABOLA	parabola-rectangular-type law of the BPEL, as per Annex 14 on HP concretes	ε_u [, af_ε] [, af_σ]
09	EC02	SARGIN	a Sargin-type law in Eurocode 2, in which af_ε applies as a denominator to the Young's modulus tangential to the origin: $E_c = 1.05E_{cm}/af\_\varepsilon$. E_{cm} is the current Young's modulus.	ε_0, ε_u, σ_t [, af_ε] [, af_σ]
10	EC02	PARABOLA	parabola-rectangular-type law in the ENV version of Eurocode 2 Consequently, this law is obsolete.	ε_u, σ_t [, af_σ]
11	EC08	CONCRETE	Confined concrete law defined in Annex E of EC8: EN 1998-2:2006.	σ_0, ε_0, ε_u
12	PARABOLA	SIMPLE	Simple parabola law. The current Young's modulus is the module at the origin.	σ_0, ε_u, σ_t
13	S_BRANCHE	SIMPLE	Sargin's law with a linear descending branch bounded by ε_u, σ_u.	ε_0, ε_u, σ_u, σ_t, k_1, k_2
14	SARGIN	SIMPLE	The general Sargin law with a non-linear descending branch.	ε_0, ε_u, σ_t, k_1, k_2
15	STEEL	BILINEAR	behavior law of a bilinear steel	σ_0, ε_u, σ_u
16	STEEL	BA83	behavior law of a steel that complies with BAEL 83	σ_0, σ_u

Table 8.1 - Base material: elastoplastic behavior laws

NOTE: All these laws use the Young's modulus $para\_supp_1$ defined in the MATERIAU commands selected by part_materiaux.

Functions

This command is used to add the parameters describing the elastoplastic behavior to the definition of a group of base materials.

Conditions of use

- In the absence of a command of this type, all the base materials are assumed to have perfect linear behavior.
- This optional command must be introduced in CALCUL NON LINEAIRE mode, before any construction commands, with one command per group of base materials for which the elastoplastic behavior is to be taken into consideration.
- It is forbidden to redefine the elastoplastic characteristics of a material.

Methodological advice

- When the "height" of a section is "small" in a given direction, since the value of the discretization interval set by default is excessive, it must be redefined by taking an order of magnitude of 1/20 of the height in question.
- In a beam with non-linear materials, the transition elements between two sections of different types or different topologies must used an elastic material. This material must be referenced in the geometry module. It must be identical to the material of the adjacent elements, but its name must not have the same particle as the non-linear material. The linear material must be allocated to the transition elements in the geometry file.

Examples

```
CALCUL  NONLINEAIRE
..............
$ for the elements made of materials whose names contain the particle P1
$ and that have an elastoplastic behavior that complies with the BPEL
$ and are of a parabolic-rectangular type, the number of calculation
$ sections is 3 by default, the discretization of the sections is
$ fixed at 0.05 by default in the directions Gy and Gz and the
coefficients
$ of affinity are fixed at 1.0 by default for the epsilon and sigma axes
$ of the deformation-stress diagrams
MATERIAUX  ELASTOPLASTIQUES  P1
  TYPE  BPEL  SOUS_TYPE  PARABOLE
  EPSU  0.0035
$ for this group of analogous elements, the number of calculation
sections
$ and the discretization are explicitly defined
MATERIAUX  ELASTOPLASTIQUES  P2
  TYPE  BPEL  SOUS_TYPE  PARABOLE
  EPSU  0.0035
  SECTIONS  7
  PASY  0.01    PASZ  0.01
..............
```

Related commands

UNITES ; CALCUL NONLINEAIRE

8.11 – ELASTOPLASTIC STEELS

ACIERS ELASTOPLASTIQUES part_aciers

$$\text{TYPE}\ \left\{\begin{array}{l} \text{LINEAIRE}\quad \text{SOUS\_TYPE}\ \left\{\begin{array}{l}\text{PARFAIT}\\ \text{FRAGILE}\end{array}\right\} \\ \\ \text{ELASTOPLASTIQUE}\quad \text{SOUS\_TYPE}\ \left\{\begin{array}{l}\text{PARFAIT}\\ \text{BILINEAIRE}\\ \text{RAFFERMISSEMENT}\end{array}\right\} \\ \text{POLYNOMIAL}\quad \text{SOUS\_TYPE}\ \left\{\begin{array}{l}\text{THURLIMANN}\\ \text{BA83\_ECROUI}\\ \text{BPEL\_ACTIF}\end{array}\right\} \end{array}\right\}$$

[SIGL σ_l] [SIGE σ_e] [EPSU ε_u] [SIGU σ_u]

[EPS1 ε_1] [EPS2 ε_2] [AMOD af_module]

Parameters

- part_aciers: selection particle (similar to a name). All the types of steels defined by the CARACTERISTIQUES CABLES commands of the GE1 module (see Chapter 3) representing a group of passive or active steels, and whose names contain this particle are affected.
- σ_l: stress at the end of the linear part of the diagram representing their behavior law $\sigma(\varepsilon)$ (Figures 8.10 and 8.11).
- σ_e: conventional elastic limit.
- ε_u, σ_u: ultimate deformation, or the rupture and rupture stress.
- ε_1, ε_2: ultimate deformation of the first plateau and initial deformation of the second plateau, of the diagram in question with strengthening.
- af_module: coefficient of affinity to be applied to the stress deformation diagram, in the axis tangential to its origin. It must be positive and equal 1.0 at the most (default value).

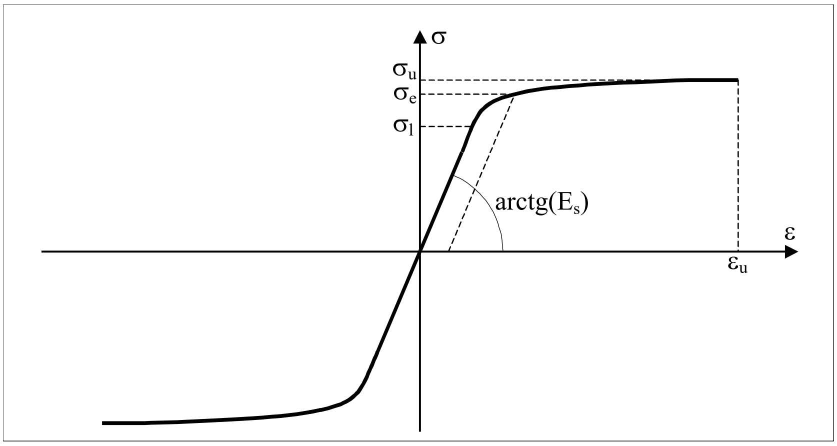


Figure 8.18 - Steels: diagram of of typical stress deformations

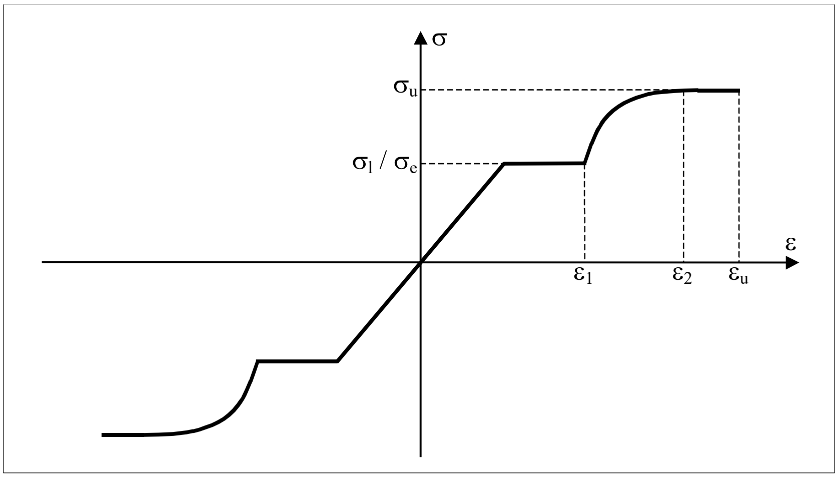


Figure 8.19 - Steels: diagram of of stress deformations with strengthening

The table below shows the parameters to be provided, according to the type and the subtype of the selected behavior law.

No.	Type	Subtype	Description	Parameters
01	LINEAR	PERFECT	perfect linear behavior law without rupture limit	none
02	LINEAR	FRAGILE	fragile linear behavior law with rupture limit	σ_l [, af_module]
03	ELASTOPLASTIC	PERFECT	perfect behavior law with plastic plateau	σ_l, ε_u [, af_module]
04	ELASTOPLASTIC	BILINEAR	bilinear behavior law with any linear plastic zone	σ_l, ε_u, σ_u [, af_module]
05	ELASTOPLASTIC	STRENGTHENING	behavior law of natural steels	σ_l, ε_u, σ_u, ε_1, ε_2 [, af_module]
06	POLYNOMIAL	THURLIMANN	Thürlimann's law (simplified form of law 7)	σ_l, ε_u, σ_u [, af_module]
07	POLYNOMIAL	BA83_ECROUI	behavior law of a hard-tempered steel that complies with BAEL 83	σ_l, σ_e, σ_u [, af_module]
08	POLYNOMIAL	BPEL_ACTIF	behavior law of an active steel that complies with the BPEL	σ_l, σ_e, σ_u [, af_module]

Table 8.2 - Steels: elastoplastic behavior laws

Functions

This command is used to add the parameters describing the elastoplastic behavior to the definition of a group of steel-types.

Conditions of use

- In the absence of a command of this type, all the steel-types are assumed to have perfect linear behavior.
- This optional command must be introduced in CALCUL NON LINEAIRE mode, before any construction commands, with one command per group of steel-types for which the elastoplastic behavior is to be taken into consideration.
- It is forbidden to redefine the elastoplastic characteristics of a steel-type.

Examples

```
CALCUL  NONLINEAIRE
..............
$ elastoplastic charactersitics of the passive steels of beam 1
$ (identified by particle P1), compliant with BAEL 83 (hard-drawn steels)
$ the coefficient of affinity parallel to the tangent at the origin is
1.0
$ by default
ACIERS  ELASTOPLASTIQUES  P1
TYPE  POLYNOMIAL  SOUS_TYPE  BA83_ECROUI
SIGL  (0.7*48930.0)
SIGE  48930.0
SIGU  57230.0
$ same characteristics for beam 2 (particle P2)
ACIERS  ELASTOPLASTIQUES  P2
TYPE  POLYNOMIAL  SOUS_TYPE  BA83_ECROUI
SIGL  (0.7*48930.0)
SIGE  48930.0
SIGU  57230.0
..............
```

Related commands

UNITES ; CALCUL NONLINEAIRE

8.12 - TITLE

TITRE titre_page

Parameters

- titre_page: title that will be reproduced at the top of each page of the results of the PH3 module, outside the command echo (character string).

Functions

As long as it does not come across a TITRE command, the PH3 module choose the main title of the model as the title of the page (the first TITRE command of the PH1 module). This command redefines the current page title, which continues to apply until another TITRE command replaces it.

Conditions of use

- ❐ This optional command can be introduced several times to identify the main construction steps or groups of commands.
- ❐ The title of the current page is also used to identify a structure saved using the SAUVER command or a report recorded using the ETAT command.

Examples

```
PHASES  DE CONSTRUCTION
$ preliminary step of the simulation, for group 1 of commands
$ the page title is the main title of the model (default option)
$ group 1 of commands
$ update of the page title for group 2 of commands
TITRE  '*** VIADUC D''ACCES B, CONSTRUCTION DU FLEAU 1 ***'
$ group 2 of commands
...........
$ update of the page title before saving an ETAT
TITRE  'ETAT PROBABLE EN FIN DE CONSTRUCTION DU FLEAU 1'
ETAT  101
$ update of the page title for group 3 of commands
TITRE  '*** VIADUC D''ACCES B, CONSTRUCTION DU FLEAU 2 ***'
$ group 3 of commands
...........
$ update of the page title before saving an ETAT
TITRE  'ETAT PROBABLE EN FIN DE CONSTRUCTION DU FLEAU 2'
ETAT  102
$ update of the page title for group 4 of commands
TITRE  '*** VIADUC D''ACCES B, COULAGE DES PARTIES SUR CINTRE, CLAVAGE
***'
$ group 4 of commands
...........
$ update of the page title before saving a structure
TITRE  'ETAT PROBABLE EN FIN DE CONSTRUCTION'
SAUVER  FINCONST
FIN
```

Related commands

EDITER / NONEDITER ; SAUVER ; ETAT ; IMPRIMER EXTREMAS;

IMPRIMER TENSIONS

8.13 - CHAINS

CHAINETTES [SIMPLIFIEES] [ITERATION nb_max_iter] nb_chainettes

$\langle \text{no\_element} \quad \text{W} \rangle_{\text{nb\_chainettes}}$

Parameters

- nb_max_iter: the maximum number of iterations of the equilibrium process
- nb_chainettes: total number of biarticulated elements, declared as a "chain" or "stay", positive.
- no_element, W: number of an element and corresponding weight per linear meter, which includes that of the resisting part, and that of a possible protective envelope.

Functions

This command completes the definition of the biarticulated elements that must be considered as chains. There are two options to model them:

- in "chain calculation" mode, which takes account of their behavior as chains and the actual Young's modulus. This mode is chosen by default.
- in "simplified chain" mode they behave like biarticulated elements whose dead weight is deduced from the values attributed to the W parameter.

Chain calculation mode

In this mode, the exact behavior of a stay is simulated by an iterative process that converges when the entire structure is in equilibrium. The forces at its extremities are calculated according to:

- its section, its actual Young's modulus and its weight per linear meter,
- the exact position of its extremities,
- its length when unladen or the tension at one of its extremities.

The thermal expansion of an element covered by this calculation mode (explicit loading) is applied by correcting its unladen length.

Simplified chains mode

In this mode, the half of the dead weight of the biarticulated element that models a stay is applied to each of its extremities when it is tensioned.

Conditions of use

- ❐ Before using this command, biarticulated elements must have been define thanks to the SECTION TYPE command of the PH1 modulus (Chapter 6), by setting all parameters to 0, except the surface and and the surface/outer perimeter ratio (recognition criterion).
- ❐ A single command of this type, inserted before any construction commands, must be used to describe all the chain type elements. It can apply to a subassembly of the biarticulated elements.
- ❐ Biarticulated elements that are not declared as chains keep their rigidity (in traction or compression).

- The SIMPLIFIEES option is forbidden in CALCUL NON LINEAIRE mode.

Methodological advice

- A very short stay that is not likely to stretch can be omitted from the list of chains.
- A long stay must normally be considered to have the behavior of a chain. However, in a first approximation, it can be considered to have a simplified behavior.

Examples

```
CHAINETTES   8
205 0.060    210 0.060    215 0.060    220 0.060
305 0.060    310 0.060    315 0.060    320 0.060
```

```
CHAINETTES   SIMPLIFIEES   8
205 0.060    210 0.060    215 0.060    220 0.060
305 0.060    310 0.060    315 0.060    320 0.060
```

Related commands

UNITES ; EDITER / NONEDITER ; ACTIVER ELEMENTS ; TENDRE ELEMENTS

CAS DE CHARGE ; CHARGEMENT [IDENTIQUE] NOEUDS / EXTREMITES

8.14 – DEFINE PUSHED STRUCTURE

DEFINIR STRUCTURE POUSSEE nom_strpou nb_pou_ele

$$[\begin{Bmatrix} \text{TRANSLATIO N} \quad [\begin{Bmatrix}+\\-\end{Bmatrix}]\text{element\_1} \quad [\begin{Bmatrix}+\\-\end{Bmatrix}]\text{element\_2} \\ \\ \text{ROTATION} \quad \text{x\_axe} \quad \text{y\_axe} \quad \text{z\_axe} \quad \theta_1 \quad \theta_2 \quad \theta_3 \quad \delta_\theta \\ \text{EMBOITEMEN T} \end{Bmatrix}]$$

$$* \begin{Bmatrix} [\text{POUTRES} \quad \text{nb\_poutres} \\ \langle \text{no\_poutre} \rangle_{\text{nb\_poutres}}] \\ [\text{ELEMENTS} \quad \text{nb\_element s} \\ \langle \text{no\_element} \rangle_{\text{nb\_elements}}] \end{Bmatrix}$$

Parameters

The line breaks in the above list of beams and elements are compulsory.

- nom_strpou: the name given to a pushed structure.
- nb_pou_ele: the total number of beams and elements that make up the pushed structure, positive.
- element_1, element_2: number of two elements belonging to the pushed structure, whose designated extremities carry the straight line indicating the direction of translation, if the structure is pushed by TRANSLATION. The *minus* sign designates the origins of the elements and the absence of a sign or the *plus* sign designates their extremities. The two terms must have the same sign. The translation is in the direction element_1 => element_2.
- x_axe, y_axe, z_axe: the coordinates in the global reference frame of any point of the axis of rotation, if the structure is pushed by ROTATION.
- θ_1, θ_2, θ_3: standardized triple rotation (see Figure 1.2) orienting a local reference frame, whose z axis is parallel to the axis of rotation, relative to the global reference frame (zero angles when these reference frames are the same).
- δ_θ : angle of rotation of the pushed structure, when it is displaced from an element in the positive direction of the push (see POUSSER [AUTOMATIQUEMENT] STRUCTURES command).
- nb_poutres: the number of beams belonging to the pushed structure, positive, if the POUTRES option is used.
- no_poutre: number of a beam.
- nb_elements: the number of elements belonging to the pushed structure, positive, if the ELEMENTS option is used.
- no_element: number of an element. The incidences are designated by the numbers of the elements that occupy them when the model is built (initial incidences).

The pushed structure can be exclusively made up of beams or elements.

Functions

A "push path" is defined as a series of elements that replace one another when they are pushed.

This command defines a pushed structure, relative to a push path, that can be subsequently called by the POUSSER [AUTOMATIQUEMENT] STRUCTURES command.

A pushed structure is an ordered list of numbers of elements that, in a push operation, form the entire envelope of the incidences of the path and/or the initial incidences of a series of elements, associated with a type of push.

The type of push is defined by the TRANSLATION, ROTATION or EMBOITEMENT options.

- DROIT push (default option) applies to structures whose rectilinear mean fibers make up the push paths,
- push by TRANSLATION applies to structures with non-rectilinear mean fibers, pushed on rectilinear paths,
- push by ROTATION applies to structures with circular mean fibers, and/or pushed on circular paths,
- push by EMBOÎTEMENT applies to structures with helical mean fibers, and/or pushed on helical paths.

The list of beams and/or elements must be compiled according to the following rules:

- all the incidences crossed in the push operation must be in the list, and in particular those of the zone of prefabrication,
- the initial incidences of the pushed elements must be introduced, even if they are not crossed. This is true of the incidences of a cut-water,
- the numbers of the elements in the list must be introduced in an order so that, when an element is pushed in the positive direction, the element on the incidence in position i occupies the incidence in position i+1, irrespective of the value of i,
- if the internal direction of movement of certain beams does not match the order in which their elements are introduced in the list, the corresponding incidences must be described as elements.

Conditions of use

- ❐ The general mechanical model must respect the constraints imposed for the modeling of pushed structures (see Annex D).
- ❐ This optional command must be introduced before the first POUSSER [AUTOMATIQUEMENT] STRUCTURES that uses it.
- ❐ This command cannot be introduced in CALCUL NON LINEAIRE mode or in CONCORDANCE SUSPENDUE mode.
- ❐ The total number of pushed structures is limited to 10.
- ❐ The name of a pushed structure that is already in use cannot be used again.
- ❐ An element cannot belong to different pushed structures.
- ❐ Geometric checks are made with a tolerance of 0.01 m.

Methodological advice

- See Annex D, modeling pushed structures and simulation of pushes.

Examples

```
$ pushed structure with medium rectilinear fiber made up of a beam
$ comprising its prefabrication zone, and a cut-water of nine elements
$ type of push (unspecified) straight, by default
DEFINIR  STRUCTURE  POUSSEE  TABPOU  10
POUTRE  1
101
ELEMENTS  9
251 A 259

$ push structure analogous to the above, with helical medium fiber
$ type of push specified, by nesting
DEFINIR  STRUCTURE  POUSSEE  TABPOU  10
EMBOITEMENT
POUTRE  1
101
ELEMENTS  9
251 A 259

$ beam with non-rectilinear mean fiber, linked to a rectilinear
$ push line by transverse rigid elements (figure D.x)
$ two pushed structures to be defined, because there are two push paths
$ type of push specified, by translation; direction
$ indicated by the end nodes of two homologous elements
$ ---------------------------------------------------------------------
$ upper structure comprising longitudinal elements:
$ deck (including the prefabrication zone) and cut-water
DEFINIR  STRUCTURE  POUSSEE  TABPOU  10
TRANSLATION  1 201
POUTRE  1
101
ELEMENTS  9
251 A 259
$ lower structure comprising transverse link elements
DEFINIR  STRUCTURE  POUSSEE  LIAISONS  260
TRANSLATION  210 410
ELEMENTS  260
210 A 469

$ structure analogous to the above, support line on a circle with a
$ vertical axis passing through the point [100.0, -100.0, 0.0],
$ positioning angles of the local coordinates, in which z is the axis of
zero rotation
$ elementary rotation of one degree in the negative direction
$ replace in the example above (two occurences):
$     TRANSLATION  ...
$ by ROTATION     100.0 -100.0  0.0    0.0 0.0 0.0    -1.0
```

Related commands

CALCUL NONLINEAIRE ; SUSPENDRE / CONTINUER CONCORDANCE

PLACER ELEMENTS ; POUSSER [AUTOMATIQUEMENT] STRUCTURES

8.15 – FORM TRAVELER FOOTING

EMPATTEMENT EQUIPAGES nb_max_elements

Parameters

- nb_max_elements: the number of elements that determines the maximum footing of form travelers to be installed or moved. Greater than or equal to 1.

Functions

This command defines the maximum footing of all the form travelers used after it. It continues to apply until a new command of this type is redefined.

The footing of form travelers is the number of elements demarcating the longitudinal footprint of its zone of action behind and in front of its current positioning element (Figures 6.2, 8.12 and 8.15). The form travelers must not take any footing outside this zone, nor support elements outside this zone.

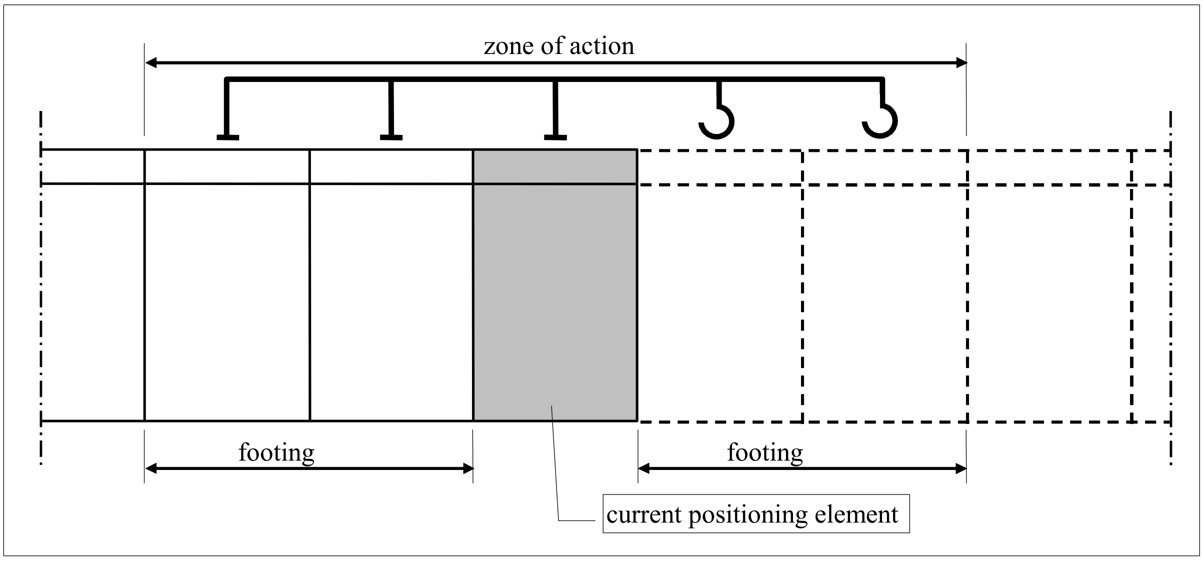


Figure 8.20 - Form travelers with a two-element footing

Fixing the maximum footing allows us to:

- demarcate the search zones attached to the elements supporting the form travelers when they are installed or moved, and check that all their supports are inside these zones,
- check that all elements suspended from form travelers are located inside their zones of action.

Conditions of use

- ☐ As long as the footing has not been defined, it is set to 1 by default.
- ☐ If the maximum footing is greater than 1, this command must be introduced before the first PLACER EQUIPAGES command that uses it.

Examples

```
$ first group of commands using a maximum overall base width
$ fixed by default to an element
PLACER  EQUIPAGES  ...
............
PLACER  ELEMENTS  COULES  SUSPENDUS  ...
............
$ maximum overall base width redefined with two elements
EMPATTEMENT  EQUIPAGES  2
$ continuation of the construction
............
```

Related commands

PLACER EQUIPAGES ; DEPLACER EQUIPAGES ; PLACER ELEMENTS

8.16 – PRESTRESS LOSSES

PERTES PRECONTRAINTE $\left\{\begin{matrix}\text{IP2}\\\text{BPEL}\\\text{EC2}\end{matrix}\right\}$

Functions

This command definitively designates the reference regulation (IP2, option IP2, BPEL, option BPEL or EC2, option EC2) for:

- the calculation of the losses of tension in the cables due to the creep and shrinkage of the concrete and the relaxation of the steels, in CALCUL FORFAITAIRE mode, as per IP2 and BPEL only (the flat-rate law of EC2 is not encoded),
- the pure relaxation law of the prestressing steels, in CALCUL RHEOLOGIQUE FIN mode, according to the three options.

Conditions of use

- This command is compulsory if the viscoelastic behaviors of the concrete and the steels are taken into consideration. A single command must be placed before:
  - the AFFECTER CONTRAINTES command (if the PH3 module is working in CALCUL FORFAITAIRE mode),
  - the first DATE command (if the PH3 module is working in CALCUL RHEOLOGIQUE FIN mode).

Examples

```
PHASES $ work in CALCUL FORFAITAIRE mode
$ losses of tension in cables due to creep and retraction of the concrete
$ and the relaxation of the steels calculated as per the BPEL
PERTES  PRECONTRAINTE  BPEL
$ other commands to apply options
$ start of the construction
AFFECTER  CONTRAINTES  UNIQUES  600.0
............
FIN
```

```
PHASES $ work in RHEOLOGIQUE FIN mode
$ pure relaxation law of the prestress steels
$ set by the BPEL
PERTES  PRECONTRAINTE  BPEL
$ other commands to apply options
$ start of the construction
DATE  0
............
FIN
```

Related commands

AFFECTER CONTRAINTES ; DATE

8.17 – OPTIMIZE NUMBERING

$$\text{OPTIMISER} \quad \text{NUMEROTATION} \left\{ \begin{array}{l} \left\{ \begin{array}{l} \text{MAFF} \\ \text{MAFC} \end{array} \right\} \left\{ \begin{array}{l} \text{DEGRE\_MINIMUM} \\ \text{PSEUDO\_PERIPHERIQUE} \\ \text{IMPOSE} \quad \text{no\_noeud} \end{array} \right\} \\ \text{IDENTIQUEMENT} \\ \text{REPRISE\_PRECEDENTE} \end{array} \right\}$$

Parameters

- no_noeud: number of the node where the renumbering starts, if it is imposed.

Functions

This command definitively determines the *internal* renumbering mode of the nodes in the model. This renumbering is important, because it conditions the size of the matrix of general rigidity and, consequently, the speed at which most calculations are executed.

The *vicinity* of a node is defined as all the nodes that are linked to it by elements (adjacent nodes). The *front* is the union of the vicinities of the renumbered nodes, after deducting all the renumbered nodes. The width of the front equals its number of nodes.

The renumbering options are:

- MAFF: indicates that we are using the method of minimization of the increase in the width of the front, with a search for the nodes in the front. This method usually quickly produces "good" renumbering,
- MAFC: indicates that we are using the method of minimization of the increase in the width of the front, with a search amongst the nodes that have not been renumbered. This method produces "very good" renumbering, but must only be used for complex and critical structures in terms of calculation time,
- IDENTIQUEMENT: indicates that the numbering imposed by the PH1 module when the model was built remains unchanged. This is an order of arrangement that cannot be controlled by data.
- REPRISE_PRECEDENTE: the numbering from the previous phasing is kept even after the PH1 module has been launched provided that the number of nodes of the model is the same.

With the MAFF and MAFC options, it is necessary to designate a starting node that launches the optimization process, using one of the following additional options:

- DEGRE_MINIMUM: indicates that the starting node is the first node encountered with a minimum of adjacent nodes,
- PSEUDO_PERIPHERIQUE: indicates that the starting node is a "pseudo-peripheral" node, i.e., a topologically extreme node,
- IMPOSE: indicates that the starting node is explicitly designated by its number.

Conditions of use

- In the absence of this optional command, the PH3 module optimizes the internal renumbering of the nodes using the default options: MAFF DEGRE_MINIMUM.
- If this command is provided, its unique occurrence must be before any construction commands.
- When the model is not connected, the renumbering algorithms no longer work and the use of an OPTIMISER NUMEROTATION IDENTIQUEMENT command is compulsory.
- The results of the renumbering are only visible if the CARTOUCHES option of the EDITER command has been activated beforehand.
- The REPRISE_PRECEDENTE option is useful when the user modifies in the PH1 module only the characteristics of the sections and keeps the same form of structure. This then makes it possible to eliminate the identical renumbering of the nodes when it is expensive in computation time.

Methodological advice

- This command must only be used on complex structures and construction phases, because the default option is optimal in most cases.
- The MAFF PSEUDO_PERIPHERIQUE options produce a better result than the default options, but take more calculation time.
- The starting node must only be imposed in very rare cases.
- The REPRISE_PRECEDENTE option should be used exclusively for the cases provided above.

Examples

```
PHASES
............
$ optimize the internal renumbering of the nodes according to the
$ method of minimisation of the increase in the width of the front
$ search for nodes amongst those that are not renumbered
$ pseudo-peripheral start node
OPTIMISER  NUMEROTATION  MAFC  PSEUDO_PERIPHERIQUE
............
$ start of the construction

DATE  0
............

PHASES
............
$ do not renumber the nodes and keep their order of arrangement
$ when defining the model by the PH1 module, as the internal numbering
OPTIMISER  NUMEROTATION  IDENTIQUEMENT
............
$ start of the construction
DATE  0
............
```

Related commands

VERIFIER ; EDITER / NONEDITER

8.18 – CALCULATE STRESSES

CALCULER CONTRAINTES

Functions

By default, the PH3 module does not calculate the normal and tangential stresses in the sections of the beams at the points specified in their definition.

This command causes an irreversible switch to the CALCUL CONTRAINTES mode, for which the PH3 module calculates, when a change is applied, the variations in stress and the new state of stress in the elements of the activated beams.

Conditions of use

- ❐ If this optional command is provided, its unique occurrence must be before any construction commands.
- ❐ This command must be introduced before any CALCULER EXTREMAS CONTRAINTES and EDITER … CONTRAINTES commands.

Examples

```
PHASES
............
CALCULER  CONTRAINTES
CALCULER  EXTREMAS  CONTRAINTES  ...
............
EDITER  ...  CONTRAINTES  ...
............
$ start of the construction
DATE  0
............
............
FIN
```

Related commands

CALCULER EXTREMAS ; ENREGISTRER ; EDITER / NONEDITER

8.19 – CALCULATE EXTREME VALUES

CALCULER EXTREMAS *etudes*

COMPOSANTE no_cpp [CONCOMITANTE $\langle \text{no\_cpc} \rangle_{\text{nb\_cpc}}$]

In this label, replace *etudes* by:

$$\left\{ \begin{array}{l} \text{REACTIONS} \\ \text{DEPLACEMENTS} \\ \text{EFFORTS} \left\{ \begin{array}{l} \text{ELEMENTS} \\ \text{SECTIONS} \end{array} \right\} \\ \text{CONTRAINTES} \left\{ \begin{array}{l} \text{NORMALES} \\ \text{TANGENTES} \end{array} \right\} \end{array} \right\}$$

Parameters

- no_cpp, no_cpc: number of a principal study component, and an associated concomitant component,

The different types of results that can be used to calculate extreme values, their numbered components and the corresponding abbreviations are described in Table 1.1.

Functions

This command tells the PH3 module to keep, for the rest of the construction, the extreme values of the ETATS of REACTIONS, EFFORTS or CONTRAINTES of the activated structure, reduced to a main study component and a possible concomitant component.

It also designates the study effects and components (main and, possibly, concomitant) to be taken into consideration to calculate the weighted or extreme states (see the EXTREMAL and PONDERE options of the ETAT command).

Conditions of use

- This optional command, which has an irreversible effect, can be introduced several times, in various steps of the construction.
- All the effects can be studied simultaneously, with all their study components.
- A query about a study effect and a study component cannot be repeated.
- EFFORTS extremes in SECTIONS or CONTRAINTES reference frames can only be requested if the model has at least one beam.
- CONTRAINTES extremes can only be requested if the CALCULER CONTRAINTES command has already been introduced.

Methodological advice

- To calculate extreme values, insert the CALCULER EXTREMAS command before the start of the construction.
- To calculate extreme or weighted states, place the CALCULER EXTREMAS commands just before the first ETAT command (EXTREMAL or PONDERE).

Examples

```
PHASES
............
CALCULER  CONTRAINTES
CALCULER  EXTREMAS  REACTIONS  COMPOSANTE 3
CALCULER  EXTREMAS  EFFORTS  ELEMENTS  COMPOSANTE 3  CONCOMITANTE 5
CALCULER  EXTREMAS  EFFORTS  ELEMENTS  COMPOSANTE 5  CONCOMITANTE 3
CALCULER  EXTREMAS  CONTRAINTES  NORMALES  COMPOSANTE 1
CALCULER  EXTREMAS  CONTRAINTES  TANGENTES  COMPOSANTE 1  CONCOMITANTE 2
............
$ start of the construction
DATE  0
............
$ the states saved below take account of all the choices of
$ effects and components of the preceding CALCULER EXTREMAS commands
ETAT  EXTREMAL ...
ETAT  PONDERE  ...
............
```

Related commands

CALCULER CONTRAINTES ; SUSPENDRE / CONTINUER ANALYSE

POUSSER [AUTOMATIQUEMENT] STRUCTURES ; ETAT

IMPRIMER EXTREMAS

8.20 - RECORD

$$\text{ENREGISTRER} \quad \left[\left\{ \begin{array}{l} \text{EFFACER} \\ * \left\{ \begin{array}{ll} \text{DEPLACEMEN TS} & \\ \text{REACTIONS} & \\ \text{EFFORTS} & * \left\{ \begin{array}{l} \text{ELEMENTS} \\ \text{SECTIONS} \end{array} \right\} \\ \text{CONTRAINTE S} & * \left\{ \begin{array}{l} \text{NORMALES} \\ \text{TANGENTES} \end{array} \right\} \end{array} \right\} \end{array} \right\} \right]$$

Functions

This command tells the PH3 module to record, for the rest of the construction, the ETATS of DEPLACEMENTS, REACTIONS, EFFORTS and/or CONTRAINTES of the activated structure in the database. Each implicit or explicit loading command (can be accumulated) launches a recording operation. These results are viewed using the RES module.

The different types of results that can be recorded and the corresponding abbreviations are described in Table 1.1.

In the absence of any option, all the possible states are recorded.

The EFFACER option deletes all the recordings made during earlier sessions. This deletion is automatic when the ENREGISTRER commands are placed before the start of the construction.

The recording of the states can be suspended by the SUSPENDRE ENREGISTREMENT command, or resumed by the CONTINUER ENREGISTREMENT command, at any time during the construction.

Conditions of use

- This optional command, which has an irreversible effect, can be introduced several times, in various steps of the construction.
- It applies to all the implicit or explicit (can be accumulated) loading commands that follow it.
- A query about an effect cannot be repeated.
- If present, the ENREGISTRER EFFACER command must be placed at the top of the group of ENREGISTRER commands.
- The recording of the states of EFFORTS in SECTIONS or CONTRAINTES reference frames can only be requested if the model has at least one beam.
- The recording of the states of CONTRAINTES can only be requested if the CALCULER CONTRAINTES command has already been introduced.

Examples

The three groups of commands below are equivalent, if they are placed before the start of the construction.

```
ENREGISTRER  EFFACER
ENREGISTRER  DEPLACEMENTS
ENREGISTRER  REACTIONS
ENREGISTRER  EFFORTS  ELEMENTS
ENREGISTRER  EFFORTS  SECTIONS
ENREGISTRER  CONTRAINTES NORMALES
ENREGISTRER  CONTRAINTES TANGENTES
```

```
ENREGISTRER  DEPLACEMENTS  REACTIONS  EFFORTS  ELEMENTS SECTIONS
ENREGISTRER  CONTRAINTES  NORMALES TANGENTES
```

```
ENREGISTRER
```

Related commands

CALCULER CONTRAINTES ; SUSPENDRE / CONTINUER ENREGISTREMENT

8.21 - SUSPEND / CONTINUE CONCORDANCE

$$\left\{\begin{matrix}\text{SUSPENDRE}\\ \text{CONTINUER}\end{matrix}\right\}\quad \text{CONCORDANCE}$$

Functions

This command is used to suspend, then restore, the hypothesis of concordance between the element numbers and their incidences that are assigned in the construction of the model.

Initially, the PH3 module works according to the hypothesis of concordance. It must be suspended before placing elements by moving them in push operations (see DEPLACES option of the PLACER ELEMENTS command).

Conditions of use

- This optional command can be introduced several times in different steps of the construction, but it is forbidden in CALCUL NON LINEAIRE mode.
- Since an option cannot be repeated, the suspensions and restorations of concordance must alternate. The first authorized command of this type is SUSPENDRE CONCORDANCE.
- In CONCORDANCE SUSPENDUE mode, it is forbidden to use the DEFINIR STRUCTURE POUSSEE, PLACER EQUIPAGES, DEPLACER EQUIPAGES and SUPPRIMER EQUIPAGES commands. The SAUVER command can be used, but the corresponding saved structure cannot be used by the ENV module to calculate envelopes.

Methodological advice

- As soon as the concordance is assumed to have been verified, it is preferable to restore it. The PH3 module checks that it has effectively been restored and that the structure has returned to its initial geometry.

Examples

```
$ pushing of a structure, preliminary operations (concordance checked)
$ pushing operations (concordance not checked)
SUSPENDRE  CONCORDANCE
$ end of construction operations (concordance checked)
CONTINUER  CONCORDANCE
```

Related commands

CALCUL NONLINEAIRE ; DEFINIR STRUCTURE POUSSEE ; SAUVER

PLACER EQUIPAGES ; DEPLACER EQUIPAGES ; SUPPRIMER EQUIPAGES

PLACER ELEMENTS ; POUSSER [AUTOMATIQUEMENT] STRUCTURES

8.22 - TRANSFER / DO NOT TRANSFER DISPLACEMENTS / TRANSLATIONS

$$\begin{Bmatrix} \text{REPERCUTER} \\ \text{NONREPERCU TER} \end{Bmatrix} \quad \begin{Bmatrix} \text{DEPLACEMEN TS} \\ \text{TRANSLATIO NS} \end{Bmatrix}$$

Functions

This command defines the displacement calculation mode for all the construction commands used after it. It continues to apply until a new command of this type is redefined.

In DEPLACEMENTS REPERCUTES mode, with the ACTIVER ELEMENTS command, and under certain conditions, the PH3 module transmits the displacements (and rotations) of the extremities of their adjacent elements to the free extremities of the activated elements. In TRANSLATIONS REPERCUTEES mode, rotations are not included.

In DEPLACEMENTS NON REPERCUTES mode (the default option), there is no transmission and the initial displacements (and rotations) of the free extremities of the activated elements are assumed to be zero.

Conditions of use

- This optional command can be introduced several times, in various steps of the construction.
- Since an option cannot be repeated, the changes of the displacement calculation mode must correspond and alternate. The first authorized command of this type is REPERCUTER DEPLACEMENTS / TRANSLATIONS.

Methodological advice

- By way of example, the DEPLACEMENTS NON REPERCUTES mode is suited to the parts of the structure cast on falsework, or built using the cantilever method, with segments cast on the spot.
- By way of example, the DEPLACEMENTS REPERCUTES mode is suited to the parts of the structure built using the cantilever method, with prefabricated segments.
- The camber to be applied matches the displacements, when the calculation mode is carefully chosen.

Examples

```
$ construction of half-spans with prefabricated segments
REPERCUTER  DEPLACEMENTS
............
$ for the following use of parts cast on arches
NONREPERCUTER  DEPLACEMENTS
```

Related commands

PLACER ARTICULATIONS ; ACTIVER ELEMENTS

8.23 - SUSPEND / CONTINUE ACCUMULATION

$$\begin{Bmatrix} \text{SUSPENDRE} \\ \text{CONTINUER} \end{Bmatrix} \quad \text{CUMUL}$$

Functions

This command defines the mode in which the effects of all the explicit loadings that follow it are taken into consideration (CAS DE CHARGE commands). It continues to apply until a new command of this type is redefined.

In CUMUL mode (the default option), the effects of a load case influence the state of deformation and solicitation of the active structure, while in CUMUL SUSPENDU, it remains unchanged.

The effects of loadings induced by other construction operations are always processed in CUMUL mode.

Conditions of use

- ❐ This optional command can be introduced several times, in various steps of the construction.
- ❐ Since an option cannot be repeated, the suspensions and restorations of the accumulation must alternate. The first authorized command of this type is SUSPENDRE CUMUL.

Examples

```
$ no SUSPENDRE/CONTINUER CUMUL command introdued; effects of the
$ accumulated implicit and explicit loads by default in the current state
PLACER  APPUIS  1
APP1  36  6*0.0
ACTIVER  ELEMENTS  POIDS  4
534 A 537
CAS DE CHARGE
'POIDS PROPRE DES BOSSAGES'
CHARGEMENT  IDENTIQUE  NOEUDS  2
134,138
0.0 0.0 -1.0  0.0 0.0 0.0
SUSPENDRE  CUMUL
$ explicit load whose effects are not accumulated in the current state
CAS DE CHARGE
'CHARGEMENT TEMPORAIRE'
CHARGEMENT  IDENTIQUE  NOEUDS  2
134,138
0.0 0.0 -15.0  0.0 0.0 0.0
$ implicit load, always processed in CUMUL mode
TENDRE  CABLES  2
C110G C110D
CONTINUER  CUMUL
$ explicit load, whose effects are accumulated in the current state
CAS DE CHARGE
.............
```

Related commands

CAS DE CHARGE

8.24 - SUSPEND / CONTINUE ANALYSIS

$$\begin{Bmatrix}\text{SUSPENDRE}\\ \text{CONTINUER}\end{Bmatrix} \quad \text{ANALYSE}$$

Functions

In CALCUL LINEAIRE or CALCUL NON LINEAIRE, this command suspends or restores the analysis of the extreme values requested by the CALCULER EXTREMAS commands that precede it.

In CALCUL NON LINEAIRE mode, it also suspends or restores the search for equilibrium.

By default, the PH3 module performs this analysis under the effect of all the implicit loadings, and the explicit loadings introduced in CUMUL mode.

Conditions of use

- This optional command can be introduced several times, in various steps of the construction.
- Since an option cannot be repeated, the suspensions and restorations of the analysis must alternate. The first authorized command of this type is SUSPENDRE ANALYSE.
- The last implicit or explicit loading that may precede a DATE command must be analyzed.
- If necessary, the analysis must be restored before saving a structure.

Methodological advice

- Suspending the analysis is necessary when a construction phase is broken down into several operations, of which only the accumulated effects are representative of reality.
- In this case, the analysis must be suspended before executing the first operation, and restored before executing the last one.
- By way of example, this happens when the striking of a span of the construction that was built in advance is concomitant with the tensioning of its cables, whereas the arch is not introduced as a bed of supports.
- It also happens when the segments of a construction built by successive cantilever operations are activated with their weight, before the cables are tensioned, which embeds them in the parts of the spans already built.

Examples

```
CALCULER  EXTREMAS  ...
..............
$ construction of a pair of arches; without this command, their own
$ weight would develop fictive tensile stresses, the activation
$ of the elements and the tensioning of the cables being concomitant
SUSPENDRE  ANALYSE
ACTIVER  ELEMENTS  POIDS  2
119  119
$ restoration of the analysis of the extremas
CONTINUER  ANALYSE
TENDRE  CABLES  2
FL2D20  FL2G20
..............
```

Related commands

CALCUL NONLINEAIRE ; CALCULER EXTREMAS ; DATE ; SAUVER

8.25 - SUSPEND / CONTINUE RECORDING

$$\begin{Bmatrix}\text{SUSPENDRE}\\\text{CONTINUER}\end{Bmatrix}$$ ENREGISTREMENT

Functions

This command suspends or resumes the recording of the states designated by the ENREGISTRER commands that precede it.

Conditions of use

- This optional command can be introduced several times, in various steps of the construction.
- It applies to all the implicit and explicit (can be accumulated) loading commands that follow it.
- It must be preceded by at least one ENREGISTRER command.
- Since an option cannot be repeated, the suspensions and restorations of the recording operations must alternate. The first authorized command of this type is SUSPENDRE ENREGISTREMENT.

Examples

```
PHASES
..............
$ saving of the normal and tangential stresses during a part
$ of the construction between the dates 256 and 402
ENREGISTRER  CONTRAINTES  NORMALES  TANGENTES
..............
DATE  0
SUSPENDRE  ENREGISTREMENT
..............
DATE  CONSTRUCTION  256
CONTINUER  ENREGISTREMENT
.............
DATE  CONSTRUCTION  402
SUSPENDRE  ENREGISTREMENT
..............
FIN
```

Related commands

ENREGISTRER

8.26 - EDIT / DO NOT EDIT

$$\left\{\begin{array}{l}\text{EDITER}\\ \text{NONEDITER}\end{array}\right\}$$

$$\left\{\begin{array}{l}
\text{MODELE} \quad \left\{\begin{array}{l}\text{EFFETS}\\ \text{ETATS}\end{array}\right\}^{*} \left\{\begin{array}{l}\text{DEPLACEMEN TS}\\ \text{REACTIONS}\\ \text{EFFORTS}\\ \text{CONTRAINTE S}\end{array}\right\} \\
\text{POUTRE[S]} \quad \left\{\begin{array}{l}\text{EFFETS}\\ \text{ETATS}\end{array}\right\}^{*} \left\{\begin{array}{l}\text{EFFORTS}\ {}^{*}\left\{\begin{array}{l}\text{ELEMENTS}\\ \text{SECTIONS}\end{array}\right\}\\ \text{CONTRAINTE S}\end{array}\right\} \quad [\text{no\_poutre}] \\
\text{DEFORMATIO NS} \\
\text{NOEUDS} \quad [\text{nb\_noeuds}] \\
\text{ELEMENTS} \quad [\text{nb\_element s}] \\
\text{COMPLETES} \\
\text{CARTOUCHES} \\
\text{CHAINETTES}
\end{array}\right\}$$

$$[\left\{\begin{array}{l}\langle \text{no\_noeud} \rangle_{\text{nb\_noeuds}} \\ \langle \text{no\_element} \rangle_{\text{nb\_elements}}\end{array}\right\}]$$

Parameters

The parameters below designate the entities for which certain results may be edited or not.

- no_poutre: number of a beam.
- nb_noeuds, nb_elements: number of selected nodes or elements, positive. By default, all the nodes or all the active elements are selected.
- no_noeud, no_element: number of a node or an element.

Functions

This command is used to build the list of results to be implicitly edited after executing the implicit or explicit loading commands, or "list of editions".

By default, no results of this type are edited and the list of editions is empty.

It can be compiled using one or more EDITER commands with cumulative effects, then it can be restricted using NONEDITER commands that target certain parts of the structure or certain results. Its content is remanent.

The different types of results that can be be edited and the corresponding abbreviations are described in Table 1.1.

Results relating to non-numbered rigid elements cannot be edited.

The meaning of the options is as follows:

- MODELE indicates that the list of editions applies to all the active structure,
- POUTRE[S] indicates that the list of editions applies to the active parts of all the beams, if no_poutre is not provided, or otherwise to the beam no_poutre,
- NOEUDS indicates that the list of the displacements and/or reactions to be edited applies to a selection of nodes. By default, all the nodes linked to active elements are concerned.
- ELEMENTS indicates that the list of the forces and/or stresses and/or deformations to be edited applies to a selection of elements. By default, all the active elements are concerned.
- EFFETS designates the structural effects of the loadings, which can be accumulated or not,
- ETATS designates the states of the active structure after each application of cumulative loadings,
- DEPLACEMENTS designates the displacements of the nodes linked to active element and any nodes that are attached to them by rigid eccentricity elements,
- REACTIONS designates the reactions of supports,
- EFFORTS or EFFORTS ELEMENTS respectively designate the forces at the origins and the extremities of the active elements of the model, of the beams, or of the beam no_poutre, expressed in element reference frames,
- EFFORTS SECTIONS designates the forces at the origins and the extremities of the active elements of the beams or of the beam no_poutre, expressed in section reference frames,
- CONTRAINTES designates the normal and tangential stresses at the origins and the extremities of the active elements of the model, of the beams, or of the beam no_poutre,
- DEFORMATIONS designates the deformations of the active elements of the model, of the beams, or of the no_poutre, in their origin and extremity sections,
- COMPLETES designates the intermediate results of the construction commands with multiple effects (DATE, TENDRE CABLES, DETENDRE CABLES, POUSSER [AUTOMATIQUEMENT] STRUCTURES). By default, only their final results are edited,
- CARTOUCHES designates the title blocks showing the content of the construction commands. By default, they are not edited,
- CHAINETTES designates the parameters of the active chain-type elements to be edited after each application of cumulative loadings.

Queries about nodes and elements are cross-referenced with those about beams or the selected beam, where appropriate.

Conditions of use

- ☐ This optional command can be introduced several times, in various steps of the construction.
- ☐ A request to extend or restrict the list of editions cannot be repeated.
- ☐ The forces in section reference frames and the stresses can only be requested if the model has at least one beam.
- ☐ The stresses can only be edited if the CALCULER CONTRAINTES command has already been introduced.
- ☐ The DEFORMATIONS option is only authorized in CALCUL NON LINEAIRE mode.

Methodological advice

- The EDITER commands must be used with care, because the volume of editions can quickly become very high, and increases with the number of nodes or elements concerned and with the number of phases of construction during which the editions remain active.
- The EDITER command is cumulative; the information entered when the command is called is added to the information already entered. The first example below first deletes the edition of the effects of elements, then it enters the elements to which a future command EDTER will apply, and finally it enters the effects to be edited. The latters apply to the three previous elements.

Examples

```
$$ Example of the cumulative effect of the command
$ removal of the editions of the effects of elements
NONEDITER ELEMENTS
$ adds number of elements to which a future command will apply
EDITER ELEMENTS 3
569 579 589
$ adds effects to be edited
EDITER MODELE ETATS EFFORTS



$$ Example use of the command
PHASES
TITRE  'MODELE A 101 NOEUDS ET 100 ELEMENTS, CONTROLE DES EDITIONS'
CALCULER  CONTRAINTES
..............
$ empty at the start, the list of editions is drawn up and activated for
$ the first time; request for a report on the effects of displacements of
$ nodes, reactions
$ of supports and forces in element coordinates, for the entire active
$ structure
EDITER  MODELE  EFFETS  DEPLACEMENTS  REACTIONS  EFFORTS
$ analogous additional request for accumumlated states
EDITER  MODELE  ETATS  DEPLACEMENTS  REACTIONS  EFFORTS
$ this command temporarily suspends the editions
SUSPENDRE  EDITIONS
DATE  0
..............
$ this command restores the editions
CONTINUER  EDITIONS
..............
$ reprocesses the forces in the list of editions
NONEDITER  MODELE  EFFETS  EFFORTS
NONEDITER  MODELE  ETATS  EFFORTS
$ adds accumulated states of the normal and tangential stresses to the
$ list of editions and forces in a beam in element and section
coordinates
EDITER  POUTRE  ETAT  CONTRAINTES  EFFORTS  ELEMENTS  SECTIONS  1
$ removal of the results relating to all the elements
NONEDITER  ELEMENTS
$ restoration of the results relating to certain elements
EDITER  ELEMENTS  4
3 4 7 9
$ removal of the results relating to all the nodes
NONEDITER  NOEUDS
$ restoration of the results relating to certain nodes
```

```
EDITER  NOEUDS  4
13 14 17 19
SUSPENDRE  EDITIONS
..............
CONTINUER  EDITIONS
```

Related commands

VERIFIER ; CALCUL NONLINEAIRE ; TITRE ; CHAINETTES

OPTIMISER NUMEROTATION ; CALCULER CONTRAINTES

SUSPENDRE / CONTINUER EDITIONS ; DATE ; PLACER APPUIS

REMPLACER APPUIS ; SUPPRIMER APPUIS ; PLACER EQUIPAGES

PLACER ARTICULATIONS ; REMPLACER ARTICULATIONS

ACTIVER ELEMENTS ; TENDRE ELEMENTS ; SUPPRIMER ELEMENTS

TENDRE CABLES ; DETENDRE CABLES ; PERTES CABLES ; CAS DE CHARGE

POUSSER [AUTOMATIQUEMENT] STRUCTURES ; REDISTRIBUER ; SAUVER

8.27 - SUSPEND / CONTINUE EDITIONS

$$\begin{Bmatrix} \text{SUSPENDRE} \\ \text{CONTINUER} \end{Bmatrix} \quad \text{EDITIONS}$$

Functions

This command suspends or resumes the editing of the results designated by the EDITER / NONEDITER commands that precede it.

Conditions of use

- This optional command can be introduced several times, in various steps of the construction.
- Since an option cannot be repeated, the suspensions and restorations of the editing of the results must alternate. The first authorized command of this type is SUSPENDRE EDITIONS.

Examples

```
EDITER  MODELE  EFFETS  DEPLACEMENTS
..............
$ temporary suspension of editions
SUSPENDRE  EDITIONS
..............
$ restoration of editions
CONTINUER  EDITIONS
..............
FIN
```

Related commands

VERIFIER ; EDITER

8.28 – ASSIGN STRESSES

$$\text{AFFECTER} \quad \text{CONTRAINTES} \quad \left\{ \begin{matrix} \text{UNIQUES} & \text{sigma\_u} \\ \text{VARIABLES} & \text{nb\_points\_s} \end{matrix} \right\}$$

$$[\left\langle \left\langle \langle \text{sigma\_ori} \rangle_{\text{nb\_sigma\_p}} \quad \langle \text{sigma\_ext} \rangle_{\text{nb\_sigma\_p}} \right\rangle_{\text{nb\_elements\_p}} \right\rangle_{\text{nb\_poutres}}]$$

Parameters

The nb_poutres, nb_elements_p and nb_sigma_p parameters, which respectively represent the number of beams of the model, the number of elements of a beam and its number of fibers to calculate normal stress, are implicitly defined by the numbers of values provided.

- sigma_u: mean final normal stress, if its distribution is assumed to be uniform, in all the sections of the beams.
- nb_points_s: total number of points of calculation of the normal stresses in all the sections of the beams (intermediate duplicated), if their distribution is assumed to be non-uniform,
- sigma_ori, sigma_ext: final normal stress at a point of calculation of the origin and end section of a beam element. These values are entered in the order of definition of the points of calculation of the normal stresses of each section, if their distribution is assumed to be non-uniform.

Functions

This command starts a construction process in CALCUL FORFAITAIRE mode of the deferred losses of prestress, and provides the necessary final normal stresses for all the points of calculation of the beams of the model.

These values do not influence the calculations of normal stresses that may be made elsewhere.

When they are provided individually, the normal stresses are assumed to be distributed in a section of a beam according to the number of calculation points, as follows:

- if is is 1, according to a plane passing through the point representative of the value of stress introduced and parallel to that of the section,
- if it is 2, according to a plane passing through the points representative of the values of stress introduced, and such that the stress is constant along all straight lines orthogonal to the straight line joining these points,
- if it is greater than 2, according to a plane passing through the points representative of the first three values of stress introduced.

The points representative of the values of stresses introduced in a section are applied to the straight lines orthogonal to its plane and passing through the imposed points of calculation (with a suitable sign convention).

Conditions of use

- A single occurrence of this optional command must be introduced before starting the construction process.
- CALCUL FORFAITAIRE mode is incompatible with CALCUL NON LINEAIRE mode.
- The model must have at least one prestressed beam.
- The PERTES PRECONTRAINTE command must be introduced first.

Methodological advice

- See Annex D, flat-rate calculation.

Examples

A model with two beams (1 and 2) with respectively 6 and 7 generic sections and 3 and 4 fibers of calculation of the normal stresses, built in CALCUL FORFAITAIRE mode.

```
PHASES
..............
PERTES   PRECONTRAINTE  BPEL
$ case 1 the distribution of the final normal stresses required to
$ calculate the deferred losses of prestress is assumed to be uniform
AFFECTER  CONTRAINTES  UNIQUES  500.0
..............
PHASES
..............
PERTES   PRECONTRAINTE  BPEL
$ case 2 the distribution of the final normal stresses required to
$ calculate the deferred losses of prestress is assumed to be non-uniform
$ beam 1, calculation points: 5(elements)*2(origin/extremity)*3(points)
$ beam 2, calculation points: 6(elements)*2(origin/extremity)*4(points)
$ (the entered values are not realistic)
AFFECTER  CONTRAINTES  VARIABLES  78
$ beam 1
600.0 600.0 600.0         $ 3 calculation points, element 1, origin
section
600.0 600.0 600.0         $ 3 calculation points, element 1, extremity
section
24*600.0              $ other elements
$ beam 2
500.0 500.0 500.0 500.0   $ 4 calculation points, element 1, origin
section
500.0 500.0 500.0 500.0   $ 4 calculation points, element 1, extremity
section
40*500.0              $ other elements
```

Related commands

UNITES ; CALCUL NONLINEAIRE ; PERTES PRECONTRAINTE ; DATE

PLACER ELEMENTS ; ACTIVER ELEMENTS ; TENDRE CABLES

PERTES CABLES ; REDISTRIBUER ; ETAT

8.29 - DATE

$$\text{DATE}\quad [\left\{\begin{array}{ll}\text{ACTUALISER} & \\ \text{CONSTRUCTION} & \\ \text{VIEILLISSEMENT} & [\text{INFINI}]\end{array}\right\}]\quad [\text{ETAT}]\quad [\text{date}]$$

Parameters

- date: date on which it is assumed that all the construction operations that follow it are completed, until the next DATE command or the FIN command (an integer or actual number days).

The increase (not necessarily strict) of the dates must be checked when the construction takes place in the order of introduction of the commands. This is not always the case when the possibility of automatically reordering the construction commands is used, relative to the dates provided in a "partial disorder" ("TRI DES PHASES" option to be activated in the run menu of the PCP modules). The user can click on “Options”, “Phases sorting” then on “Active” in order for PCP to put the construction phases in a chronological order. The state of the option “Phases sorting” appears at the bottom of the window. If this option is activated, a file *.PHASES_TRIEES.dec (where * is the name of the data file given by the user) is created and includes the construction phases in a chronological order so that the user can check the construction.

Functions

This command is used to introduce the time variable in the form of a history of the construction operations, when the calculation of the structural effects of the deferred phenomena must be rigorously simulated. The first DATE command marks the start and fixes the date of the start of the construction process. It indicates the chosen operating mode: CALCUL RHEOLOGIQUE FIN.

The ACTUALISER option indicates that it is necessary to recalculate the Young's modulus of the active elements, if they depend on the age of their constituent materials. In the absence of the ACTUALISER, CONSTRUCTION or VIEILLISSEMENT options, they remain unchanged.

With DATE [ACTUALISER] [ETAT] date_n and $n > 1$, the interval $[\text{date}_{n-1}, \text{date}_n]$ is considered as an imposed time increment and must not exceed about 8 days.

During the period of construction, since the structure is subjected to successive loadings, the length of the time increments used to integrate the deferred phenomena must not exceed about 8 days. Beyond this limit, numerical instabilities may appear.

With the CONSTRUCTION option, the parameter date_n, and $n > 1$, the interval $[\text{date}_{n-1}, \text{date}_n]$ is considered as an imposed time increment, if its length is less than or equal to 4 days. Otherwise, it is automatically broken down into increments of the same length of between 4 and 8 days.

The aging period starts when the construction is no longer subjected to any loadings other than those due to deferred phenomena. The time increments used to integrate them can increase "very quickly", without any numerical instability appearing.

With the VIEILLISSEMENT option, the parameter date_n, and $n > 1$, the interval $[\text{date}_{n-1}, \text{date}_n]$ is automatically broken down into time increments whose length increases in geometric progression (it doubles every four increments).

The first DATE VIEILLISSEMENT $date_n$ command introduces this operating mode and fixes the start of the aging on the date $date_n$.

With the INFINI option, the "limit" date, which does not have to be provided, is obtained by adding a number of days to the date that depends on the regulation used to calculate the deferred losses of prestress (PERTES PRECONTRAINTE command).

The dates produced for the construction or the aging are added to the dates provided, to form the "integration" dates.

The CONSTRUCTION and VIEILLISSEMENT options contain the ACTUALISER option, because the Young's moduli of the materials, which depend on their age, are recalculated after each time increment has elapsed.

With the ETAT option, the elastic losses of prestress are calculated up to and including the specified date, and integrated in the probable state of the structure obtained in this way, on this date.

Here are the main processes executed for each integration date t_i, after the date of the start of the construction.

Step 1

If the cables are taut:

- calculation of the elastic losses of tension caused in these cables by the loadings applied on the date $t_{i-1,}$
- calculation of the structural effects of these losses and of the new state of deformation and solicitation of the active structure,
- calculation of the new tensions in the cables (including the viscoelastic losses),
- edition of the requested results, if the date t_i is a date provided.

The PH3 module uses this process to determine the state of the active structure on the date t_{i-1}, including the loadings and concrete-cabling interactions.

Step 2

Change of date. The new date is fixed at t_i days.

Step 3

With the ACTUALISER, CONSTRUCTION or VIEILLISSEMENT option, update of the instantaneous Young's moduli of the active elements, whose constituent materials are aging from this perspective. The rigidity of the active structure is then modified.

Step 4

If the active elements are made of materials that creep or shrink, or if the cables are taut:

- calculation of the predeformations due to the creep or shrinkage of these materials and to the relaxation of the prestressing steels, between the dates t_{i-1} and t_i ;
- calculation of their structural effects and of the new state of deformation and solicitation of the active structure,
- edition of the requested results, if the date t_i is a date provided, and if step 1 has not been completed.

Steps 2, 3 and 4 are omitted, if the dates t_{i-1} and t_i are equal.

Conditions of use

- The use of this command is only authorized if the model has at least one cable or one aging material.
- If the model has at least one cable, the PERTES PRECONTRAINTE command that determines the reference regulation used to simulate the pure relaxation of the steels, must appear before any DATE commands.
- The first DATE command must precede any construction operations.
- When several DATE commands succeed one another, without being separated by construction operations, the state of deformation and solicitation of the active structure evolves, due simply to the rheological behaviors of its aging materials.
- If necessary, the analysis of the extreme values must be restored before introducing the last implicit or explicit loading preceding a DATE command (see the SUSPENDRE / CONTINUER ANALYSE command).
- The ACTUALISER option can only be used if at least one active element is made up of a material whose Young's modulus varies over time.
- It is forbidden to return to the CONSTRUCTION option after choosing the VIEILLISSEMENT option.
- CONSTRUCTION dates or VIEILLISSEMENT dates can be mixed with dates without these options.

Methodological advice

- The DATE [ACTUALISER] [ETAT] t_i command is used when we want to impose integration dates, without the PH3 module adding any intermediate dates. In this case, it is necessary to check that the time intervals are correctly calibrated.
- The ACTUALISER option, *which is of limited interest,* can be used when certain active elements have not yet reached the age beyond which the Young's moduli of their constituent materials no longer vary.
- The CONSTRUCTION and VIEILLISSEMENT options avoid having to manage the integration intervals when selecting the dates, and update the Young's moduli of the materials, where necessary.
- Usually, the CONSTRUCTION option is taken at the start of the phasing and maintained until the entry into service. It is then definitively replaced by the VIEILLISSEMENT option.
- In addition to separating "conventional" construction operations (supports management, activation or removal of elements, loading nodes or elements, tensioning or loosening cables, etc.), the DATE command is used to:
  - edit or record a state of the structure on a given date,
  - provide a casting date for certain concrete parts (see the COULES option of the PLACER ELEMENTS command).
- When automatically reordered, all the construction phases following each DATE command are moved with it, including any control commands, for which the coherency of the new order must be verified.
- Also see Annex D, fine rheological calculation.

Examples

```
PHASES
$ strict simulation of the construction of a prestressed structure
..............
$ choice of the regulation for the calculation of deferred tension
$ in the cables due to the relaxation of the steel
PERTES  PRECONTRAINTE  BPEL
..............
$ the first DATE command starts the construction; there are no options
$ and the command sets 0 as a start date (not compulsory but often
useful)
DATE  0
PLACER  APPUIS  ...
..............
$ the systematic use of the CONSTRUCTION option provides protection
against
$ numerical instabilities; here, the interval is less than 4 days
$ and no intermediate dates are generated
DATE  CONSTRUCTION  3
ACTIVER  ELEMENTS  ...
..............
$ time interval greater than 4 days, module PH3 generates,
$ between 3 and 15 days, regularly spaced intermediate dates
DATE  CONSTRUCTION  15
TENDRE  CABLES  ...
..............
$ end of construction, the probable corresponding state is saved,
$ including the losses of elastic pre-stress calculated after 396 days
DATE  CONSTRUCTION  ETAT  396
TITRE  'ETAT PROBABLE DE L''OUVRAGE A SA MISE EN SERVICE'
SAUVER  STRUSERV
$ start of the aging period, the probable state after 1,000 days
$ is saved; same as above for the elastic losses; between
$ 396 and 1,000 days, time intervals of increasing lengths are calculated
DATE  VIEILLISSEMENT  ETAT 1000
ETAT  1
$ from now on, the CONSTRUCTION option is no longer authorized,
$ continuation of the aging process; the generated time intervals
$ continue to increase, in geometric progression
DATE  VIEILLISSEMENT  2000
..............
$ continuation and end of the aging; the time increments continue
$ to increase; the probable state at infinite time is saved
DATE  VIEILLISSEMENT  INFINI  ETAT
TITRE  'ETAT PROBABLE DE L''OUVRAGE AU TEMPS INFINI'
ETAT  2
FIN
```

Related commands

VERIFIER ; PARAMETRES RHEOLOGIQUES ; PERTES PRECONTRAINTE

SUSPENDRE / CONTINUER ANALYSE ; EDITER / NONEDITER

AFFECTER CONTRAINTES ; PLACER APPUIS ; REMPLACER APPUIS

SUPPRIMER APPUIS ; PLACER EQUIPAGES ; DEPLACER EQUIPAGES

SUPPRIMER EQUIPAGES ; PLACER ELEMENTS ; PLACER ARTICULATIONS

REMPLACER ARTICULATIONS ; ACTIVER ELEMENTS ; TENDRE ELEMENTS

SUPPRIMER ELEMENTS ; TENDRE CABLES ; INJECTER CABLES

DETENDRE CABLES ; PERTES CABLES ; CAS DE CHARGE ; SAUVER

POUSSER [AUTOMATIQUEMENT] STRUCTURES ; REDISTRIBUER ; ETAT

8.30 – PLACE SUPPORTS

PLACER APPUIS [VERINAGE [$\left\{\begin{array}{c}\text{ABSOLU}\\ \hline \text{RELATIF}\end{array}\right\}$] [FOIS nb_etapes]] nb_appuis

$$\Big\langle \text{nom\_appui} \quad \text{no\_noeud} \quad \text{exc\_x} \quad \text{exc\_y} \quad \text{exc\_z} \quad \theta_1 \quad \theta_2 \quad \theta_3$$

$$[du_x \quad du_y \quad du_z \quad d\theta_x \quad d\theta_y \quad d\theta_z]\Big\rangle_{\text{nb\_appuis}}$$

Parameters

- nb_etapes: the number of steps to perform the jacking, positive (1 by default).
- nb_appuis: the number of supports to be placed, positive.
- nom_appui: the name of a support-type, defined on an APPUI command of the PH1 module (see Chapter 6).
- no_noeud: the number of the node where it will be placed.
- exc_x, exc_y, exc_z: components of the support node translation vector in the global reference frame (zero when the support is the same as the node). Its origin is the node that may have been displaced by earlier deformation of the structure, in the absence of jacking, or the node that may have undergone imposed displacements, in the event of jacking (see VERINAGE option).
- θ_1, θ_2, θ_3: standardized triple rotation (see Figure 1.2) orienting the definition reference frame of the support-type, relative to the global reference frame (zero angles when these reference frames are the same).
- du_x, du_y, du_z, $d\theta_x$, $d\theta_y$, $d\theta_z$: vector of displacement in the global reference frame, to be imposed on the node in the event of jacking, representing its final position (ABSOLU option), or variations of displacements from its current position (RELATIF option). By default, these values are zero and applied in ABSOLU mode. They must be provided when the ABSOLU or RELATIF mode is chosen.

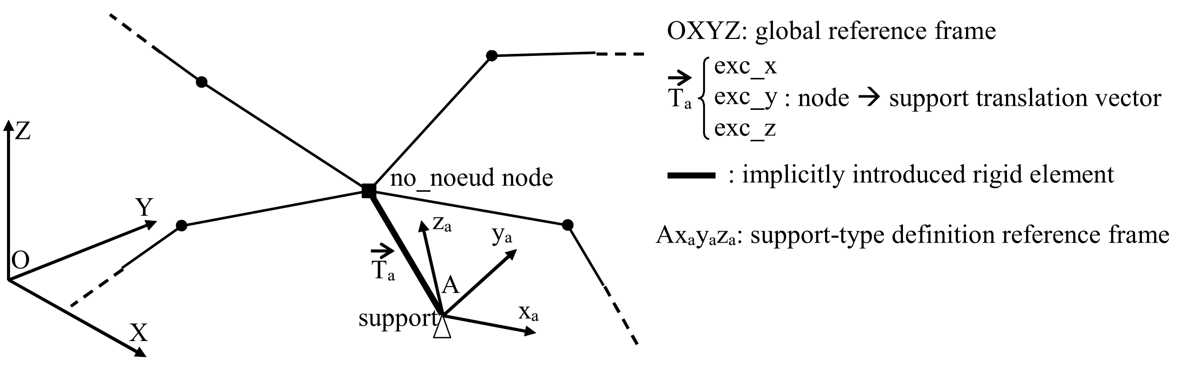


Figure 8.21 - Placing a support

Functions

This command is used to create the conditions of the supports, by assigning support-types to certain nodes that do not have any. A support node that is not yet linked to any active elements is assumed to belong to the active structure.

Each support-type can be offset relative to its destination node (which amounts to placing a rigid element between the node and the support), or reoriented in relation to the global reference frame.

Without the VERINAGE option, the future support nodes keep their positions, the rigidity of the model is modified and no loadings are applied.

With the VERINAGE option, the future support nodes are displaced beforehand by fixing their new positions of displacement variations, and the following operations are performed:

- generation of loading induced by the displacements imposed on the future support nodes,
- application of this loading to the active structure in its configuration before the installation of the supports,
- modification of the rigidity of the model by introducing the new supports,
- removal of the jacking loading by applying its opposite to the structure in its new configuration, and calculating its state of deformation and solicitation.

Conditions of use

- ☐ A node cannot receive more than one support simultaneously.
- ☐ A same support-type can be assigned simultaneously to several nodes.

Methodological advice

- ☐ The FOIS option is used, in a divergent iterative calculation, to proceed with progressive or incremental jacking, by reducing the amplitude of the applied displacements.
- ☐ Jacking in steps is useful in CALCUL NON LINEAIRE mode to achieve the convergence of the calculation processes.

Examples

The bridge deck shown below, built on an arch by advancing, is modeled using a constant-section beam that is curved in a horizontal plane.

Its supports (duplicated transversely, but modeled on a point basis) are vertically offset relative to its mean fiber, below and of the same quantity.

The x_a axes of their definition reference frame are tangential to the mean fiber.

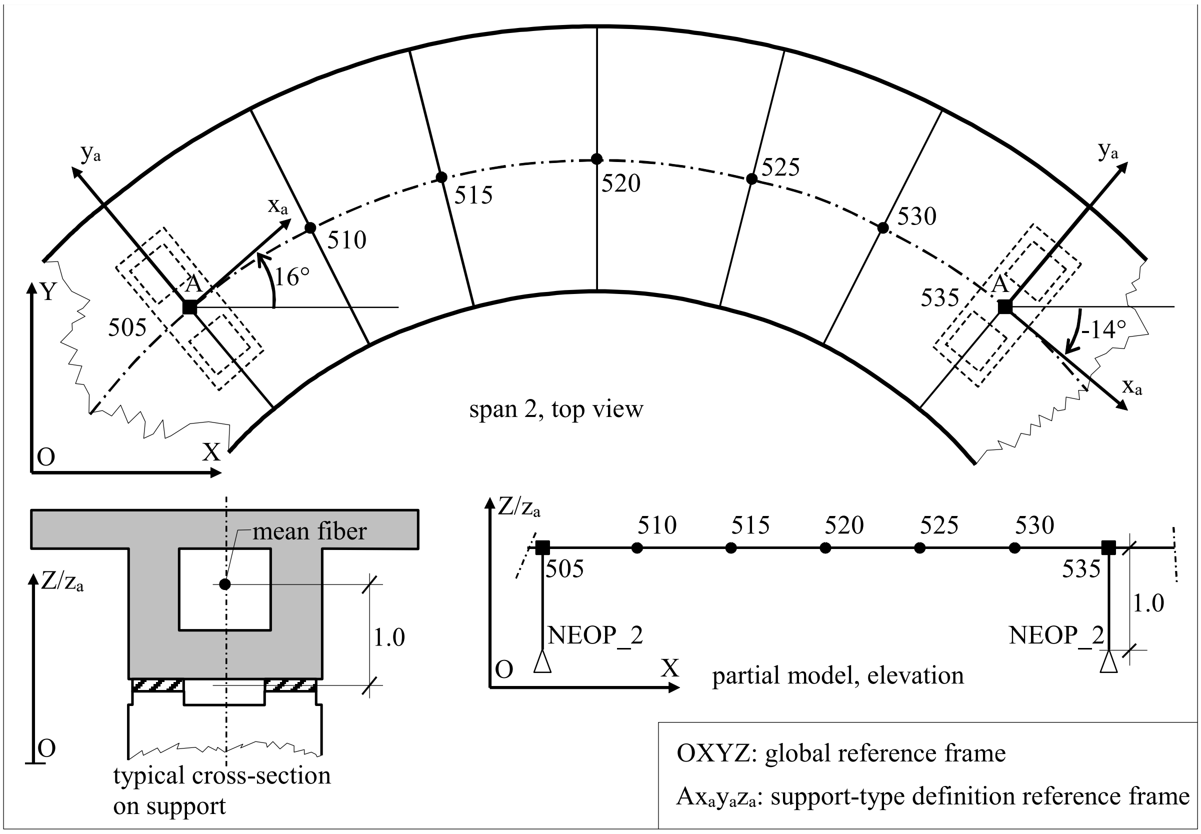


Figure 8.22 - Supports for a deck span of a curved bridge

```
$ installation of the definitive supports of span 2, vertically offset
$ per OZ by -1.0 and tangential to the mean fiber of the
$ beam; only the angle teta1 is not zero; the verinage option
$ is of no use to simulate a construction on arch by advancement
PLACER  APPUIS  2
$ support name  node n°  exc_x  exc_y  exc_z  teta1  teta2  teta3
   NEOP_2       505     0.0    0.0   -1.0   16.0    0.0    0.0
   NEOP_2       535     0.0    0.0   -1.0  -14.0    0.0    0.0
```

Related commands

VERIFIER ; CALCUL NONLINEAIRE ; EDITER / NONEDITER ; DATE

REMPLACER APPUIS ; SUPPRIMER APPUIS ; SUPPRIMER ELEMENTS

DEFORMATION APPUIS ; POUSSER [AUTOMATIQUEMENT] STRUCTURES

ETAT

8.31 – REPLACE SUPPORTS

REMPLACER APPUIS [VERINAGE $[\left\{\begin{matrix}\text{ABSOLU}\\ \text{RELATIF}\end{matrix}\right\}]$ [FOIS nb_etapes]] nb_appuis

$$\left\langle \text{nom\_appui} \quad \text{no\_noeud} \quad \text{exc\_x} \quad \text{exc\_y} \quad \text{exc\_z} \quad \theta_1 \quad \theta_2 \quad \theta_3 \right.$$

$$\left. [du_x \quad du_y \quad du_z \quad d\theta_x \quad d\theta_y \quad d\theta_z] \right\rangle_{\text{nb\_appuis}}$$

Parameters

- nb_etapes: the number of steps to perform the jacking, positive (1 by default).
- nb_appuis: the number of supports to be replaced, positive.
- nom_appui: the name of a new support-type, defined on an APPUI command of the PH1 module (see Chapter 6).
- no_noeud: the number of the node where it will replace the old support-type,
- exc_x, exc_y, exc_z: components of the new support node translation vector in the global reference frame,
- θ_1, θ_2, θ_3: standard triple rotation (see Figure 1.2) orienting the definition reference frame of the new support-type, relative to the global reference frame.
- du_x, du_y, du_z, $d\theta_x$, $d\theta_y$, $d\theta_z$: vector of displacement in the global reference frame, to be imposed on the node in the event of jacking.

The last twelve parameters follow the same conventions as their equivalents in the PLACER APPUIS command.

Functions

This command is used to modify the conditions of the supports, in certain nodes that have one, by replacing support-types and redefining their eccentricity and orientations.

Without the VERINAGE option, the following operations are performed:

- modification of the rigidity of the model by removing the old supports at the selected nodes and introducing the new supports,
- generation of loading due to the removal of the old supports (this is obtained by changing the signs of the preexisting reactions),
- application of this loading to the active structure in its new configuration, and calculation of its state of deformation and solicitation.

With the VERINAGE option, and before these operations, the PH3 module weights the diagonal coefficients of rigidity of the old support-types that are "too weak" and applies the loading induced by the displacements imposed on the support nodes, according to the degrees of freedom associated with coefficients of rigidity that are not zero.

Conditions of use

- All the selected nodes must have supports when they are replaced.
- The stability of the structure must be maintained after the replacement of the supports.

Methodological advice

- In addition to replacing a support with jacking, the VERINAGE option is used to simulate a change in the support rigidity due to creep, for example. The support-type defined with its new rigidity replaces the old one, after certain displacements of the corresponding node have been canceled.
- See also the methodological advice for the PLACER APPUIS command.

Examples

```
$ switch from the provisional support to the definitive support on pile 1
$ the VERINAGE option cancels the previous vertical settlement of the
$ provisional support, prior to the installation of the new support-type
$ because the displacements imposed on the corresponding node are zero
$ and applied in ABSOLU mode in a single step by default
REMPLACER  APPUIS  VERINAGE  1
$ support name  node n°  exc_x  exc_y  exc_z  teta1  teta2  teta3
   NEOP_1       130      0.0    0.0   -1.25   0.0    0.0    0.0
```

Related commands

VERIFIER ; CALCUL NONLINEAIRE ; EDITER / NONEDITER ; DATE

PLACER APPUIS ; SUPPRIMER APPUIS ; SUPPRIMER ELEMENTS

DEFORMATION APPUIS ; ETAT

8.32 – REMOVE SUPPORTS

SUPPRIMER APPUIS nb_appuis

$\langle \text{no\_noeud} \rangle_{\text{nb\_appuis}}$

Parameters

- nb_appuis: the number of supports to be removed, positive.
- no_noeud: number of a node whose support will be removed.

Functions

This command is used to remove the conditions of the supports, at certain nodes that have one, by performing the following actions:

- modification of the rigidity of the model by removing the supports at the selected nodes,
- generation of loading due to this removal (this is obtained by changing the signs of the preexisting reactions),
- application of this loading to the active structure in its new configuration, and calculation of its state of deformation and solicitation.

Conditions of use

☐ All the selected nodes must have supports when they are removed.

☐ The stability of the structure must be maintained after the removal of the supports.

Examples

```
$ removal of the provisional supports duplicated on piles 1 and 2
SUPPRIMER  APPUIS  4
125  135  215  225
```

Related commands

VERIFIER ; EDITER / NONEDITER ; DATE ; PLACER APPUIS

REMPLACER APPUIS ; SUPPRIMER ELEMENTS ; DEFORMATION APPUIS

POUSSER [AUTOMATIQUEMENT] STRUCTURES ; ETAT

8.33 – PLACE FORM TRAVELERS

PLACER EQUIPAGES nb_equipages

$$\left\langle \text{nom\_equipa ge} \quad [\begin{Bmatrix}+\\-\end{Bmatrix}]\text{e} \quad \text{no\_poutre} \right\rangle_{\text{nb\_equipages}}$$

Parameters

- nb_equipages: the number of form travelers to be placed, positive.
- nom_equipage: the name of the form traveler, defined on an EQUIPAGE MOBILE command of the PH1 module (see Chapter 6).
- e: order number of the beam element used, by convention, to position it.
- no_ptre: number of the corresponding beam.

The eth element of the beam no_poutre is between its generic sections e and e+1, according to the direction of its path (see Chapter 3, FIBRE REPERE command).

If e is preceded by a *minus* sign, the reference frame defining the form traveler to be placed is the generic reference frame of the section e (origin of the eth element), turned through 180 degrees around its z axis.

If it is preceded by a *plus* sign, or no sign at all, the reference frame defining the form traveler to be placed is the generic reference frame of the section e+1 (extremity of the eth element).

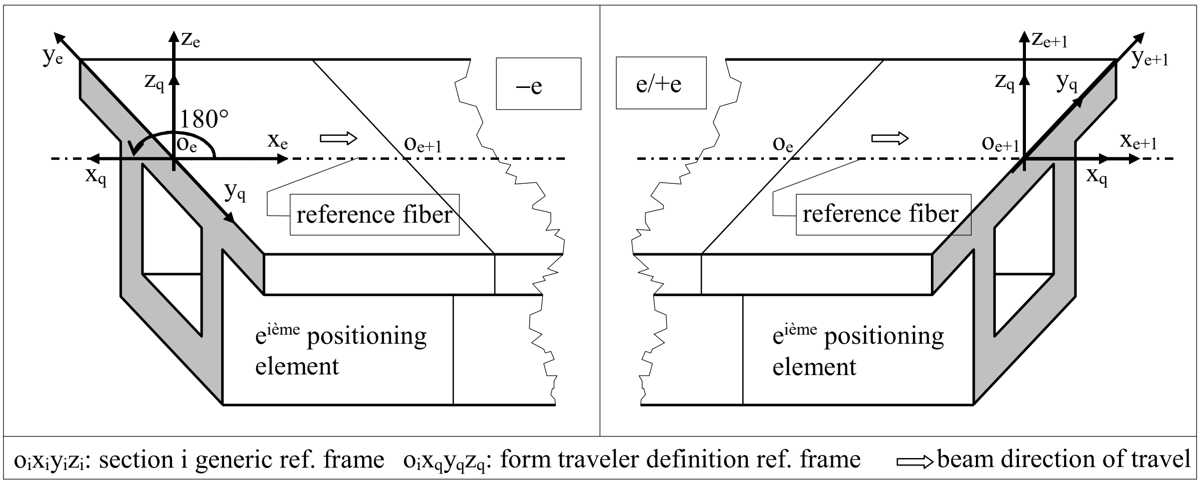


Figure 8.23 - Placing a form traveler

Functions

This command is used to place a group of form travelers on one or more beams under construction.

The weight of each form traveler passes through its system of supports and is distributed on the elements supporting it. The corresponding loading, which is applied to the active structure, can be used to calculate its new state of deformation and solicitation.

The position of each form traveler is memorized so that it can be used to install elements.

Conditions of use

- This command cannot be introduced in CONCORDANCE SUSPENDUE mode.
- Form travelers can only occupy one position at a given instant.
- All elements supporting a form traveler must be active.
- The positioning element of a form traveler can be inactive, if it does not support it.
- If a form traveler rests on more than one element or supports more than one placed element, in front of or behind its positioning element, its maximum footing must be redefined using an EMPATTEMENT EQUIPAGES command.

Examples

```
$ construction of half-span 1 of beam 101 (pile P1)
..............
$ teams EQUI_1 and EQUI_2, even if they are identical, must be
$ defined, because they are used simultaneously
$ EQUI_1 is placed on the origin section of the 36th element and returned
$ EQUI_2 is placed on the extremity section of the 38th element and is
not
$ returned; this arrangement makes the construction symmetrical
$ relative to the pile
PLACER  EQUIPAGES  2
$ team name        e      beam n°
    EQUI_1       -36      101
    EQUI_2        38      101
```

Related commands

VERIFIER ; EMPATTEMENT EQUIPAGES

SUSPENDRE / CONTINUER CONCORDANCE ; EDITER / NONEDITER ; DATE

DEPLACER EQUIPAGES ; SUPPRIMER EQUIPAGES ; PLACER ELEMENTS

8.34 – MOVE FORM TRAVELERS

DEPLACER EQUIPAGES nb_equipages

$$\left\langle \text{nom\_equipage} \quad [\begin{Bmatrix}+\\-\end{Bmatrix}]\text{e} \quad \text{no\_poutre} \right\rangle_{\text{nb\_equipages}}$$

Parameters

- nb_equipages: the number of form travelers to be moved, positive.
- nom_equipage: the name of a form traveler in place.
- e: order number of the beam element used, by convention, to reposition it (see parameter e of the PLACER EQUIPAGES command).
- no_ptre: number of the corresponding beam.

Functions

This command is used to move a group of form travelers on one or more beams under construction, by performing the following operations:

- generation of loading due to the removal of the forces of each form traveler's dead weight in its old position (obtained by changing their signs), and to the addition of the said forces in its new position,
- application of this loading to the active structure and calculation of its new state of deformation and solicitation.

The new position of each form traveler is memorized so that it can be used again to install elements.

Conditions of use

- ☐ This command cannot be introduced in CONCORDANCE SUSPENDUE mode.
- ☐ Only a form traveler that is in place and does not support any elements can be moved.
- ☐ The conditions of use of the PLACER EQUIPAGES command apply to the choice of the new position of each moved form traveler.

Examples

```
$ this example can follow the example of the PLACER EQUIPAGES command
$ the teams are shifted by one arch, symmetrically relative to
$ the pile
DEPLACER  EQUIPAGES  2
$ team name      e     beam n°       team name    e       beam n°
    EQUI_1      -37      101           EQUI_2       39      101
```

Related commands

VERIFIER ; EMPATTEMENT EQUIPAGES

SUSPENDRE / CONTINUER CONCORDANCE ; EDITER / NONEDITER ; DATE

PLACER EQUIPAGES ; SUPPRIMER EQUIPAGES ; PLACER ELEMENTS

8.35 – REMOVE FORM TRAVELERS

SUPPRIMER EQUIPAGES nb_equipages

$$\langle \text{nom\_equipage} \rangle_{\text{nb\_equipages}}$$

Parameters

- nb_equipages: the number of form travelers to be removed, positive.
- nom_equipage: the name of a form traveler in place, to be removed.

Functions

This command is used to remove a group of form travelers by performing the following operations:

- generation of the loading due to the removal of each form traveler's dead weight (obtained by changing their signs),
- application of this loading to the active structure and calculation of its new state of deformation and solicitation.

Conditions of use

- This command cannot be introduced in CONCORDANCE SUSPENDUE mode.
- A removed form traveler can be placed again.
- Only a form traveler that is in place and does not support any elements can be removed.

Examples

```
$ end of construction of half-span 1, removal of the mobile teams that
$ can be reused to build half-span 2
SUPPRIMER  EQUIPAGES  2
EQUI_1  EQUI_2
```

Related commands

VERIFIER ; SUSPENDRE / CONTINUER CONCORDANCE ; EDITER / NONEDITER

DATE ; PLACER EQUIPAGES ; DEPLACER EQUIPAGES ; PLACER ELEMENTS

8.36 – PLACE ELEMENTS

$$\text{PLACER ELEMENTS } [\left\{\begin{array}{l}\text{COULES}\\ \text{PREFABRIQUES}\end{array}\right\}] \; \textit{type\_p} \;\; \text{nb\_elements}$$

$$\left\langle [\text{nom\_equipage}] \quad \text{no\_element} \quad [[\left\{\begin{array}{c}+\\ -\end{array}\right\}]\text{na\_element}\,] \quad [\text{date\_prefa}] \right\rangle_{\text{nb\_elements}}$$

In this label, replace *type_p* by: $[\left\{\begin{array}{l}\text{SUSPENDUS}\\ \text{DEPLACES} \quad [\left\{\begin{array}{l}\text{TRANSLATION}\\ \text{ROTATION} \quad \text{nom\_strpou}\\ \text{EMBOITEMENT}\end{array}\right\}]\end{array}\right\}]$

Parameters

- nb_elements: the number of elements to be placed, positive.
- nom_strpou: name of the pushed structure to which they belong, defined on a DEFINIR STRUCTURE POUSSEE command. Its axis of rotation is used to move them by ROTATION.
- nom_equipage: name of the form traveler in place that must support the element no_element, if the SUSPENDUS option is used.
- no_element: number of an element to be placed.
- na_element: number of the element towards which it will be moved, if the DEPLACES option is used. Its sign designates the point that remains fixed, in the event of displacement by TRANSLATION or ROTATION (the *minus* signs designates its origin and the *plus* sign, or no sign at all, its extremity).
- date_prefa: date of prefabrication of the element no_element, if the PREFABRIQUES option is used. Cannot be after the current date (as set by the last DATE command).

Functions

This command is used to put a group of elements in place before they are activated, by specifying:

- their mode of implementation (cast on the spot, prefabricated), and the associated dates,
- their type of placement (direct or displacement for subsequent push operation),
- the form travelers that may be used to suspend them.

The COULES option indicates that they are all cast on the spot on the current date.

The PREFABRIQUES option indicates that every one of them was prefabricated on a given date.

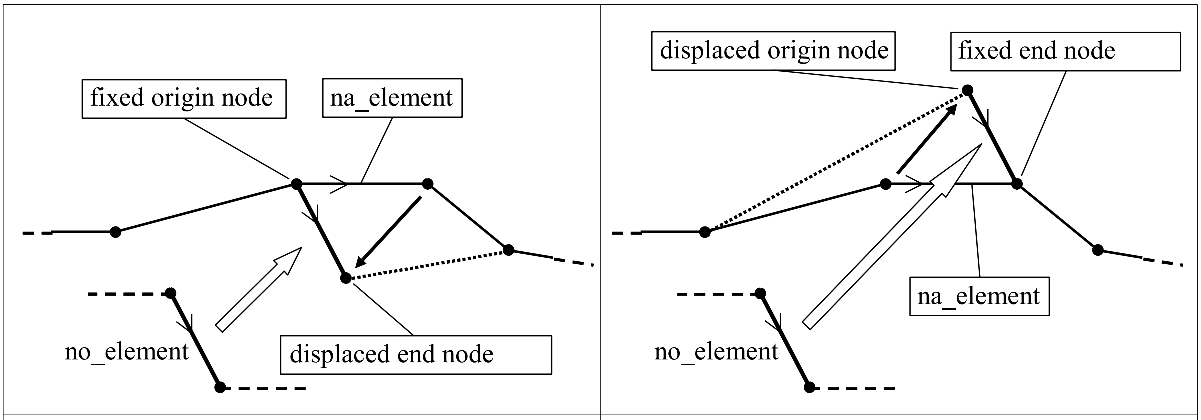
The SUSPENDUS option indicates that they are beam elements that may be suspended by form travelers. The following operations are performed:

- generation of loading due to the dead weight of the elements, transmitted to the structure by the form travelers,
- application of this loading to the active structure and calculation of its new state of deformation and solicitation.

- The displacement of suspended elements is updated in DEFORMATIONS NONREPERCUTES mode to take account of the weight of the equipment and the dead weight of the suspended elements.

With the DEPLACES option, each no_element element is displaced towards a na_element element, of which it acquires the incidence, which is different from the incidence that it occupied when the model was defined. The following conditions are met:

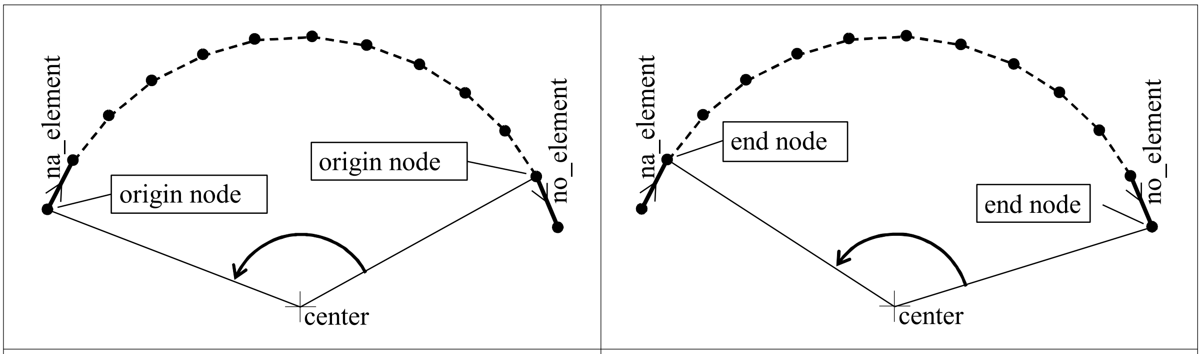
- if the DEPLACES option has no attributes, the PH3 module checks that the elements no_element and na_element have the same length and the same orientation,
- if displacement is by TRANSLATION, the no_element element is translated towards the na_element element, such that their origins coincide, if na_element is preceded by the *minus* sign, otherwise, so that their extremities coincide. When these elements do not have the same length or orientation, the incidence node of the unfixed na_element element is displaced accordingly. The PH3 module verifies that this node is free (not linked to a support or to any active elements) before recalculating its coordinates.



no_element → –na_element | no_element → na_element / +na_element

Figure 8.24 - Displacement of an element by translation

- If displacement is by ROTATION, its parameters are defined by a DEFINIR STRUCTURE POUSSEE strpou ROTATION command. The elements to be placed are turned according to the same rules that apply to translation, with regard to the fixing nodes. The PH3 module verifies that the coordinates of the connected origin or end nodes are compatible with the parameters of the rotation to be applied.



no_element → –na_element | no_element → na_element / +na_element

Figure 8.25 - Displacement of an element by rotation

- if displacement is by EMBOÎTEMENT, only the lengths of the elements must be identical. The no_element element takes the orientation of the na_element element.

Conditions of use

- This command only applies to inactive elements that are not articulated and not placed.
- If the PH3 module is not working in CALCUL RHEOLOGIQUE FIN mode, no elements can be placed with the COULES or PREFABRIQUES options.
- In CALCUL RHEOLOGIQUE FIN mode, only elements made of aging material must be placed with the COULES or PREFABRIQUES options. Their dates of use are used to determine their viscoelastic behavior.
- The SUSPENDUS option only applies to beam elements.
- A form traveler must be placed on the beam whose elements it is used to suspend.
- An element suspended from a form traveler must belong to its zone of action, which may be redefined by an EMPATTEMENT EQUIPAGES command.
- The SUSPENDUS option can only be used for elements that do not belong to a STRUCTURE POUSSEE.
- The DEPLACES option can only be introduced in CONCORDANCE SUSPENDUE mode.
- The SUSPENDUS and DEPLACES options are incompatible for a given element.

Examples

```
DATE  120
$ suspension of two beam elements cast in situ on the current date
PLACER  ELEMENTS  COULES  SUSPENDUS  2
$   team name    element n°      team name    element n°
     EQUI_1        403             EQUI_2        404

DATE  -5
$ installation of two elements whose prefabrication dates are fixed
PLACER  ELEMENTS  PREFABRIQUES  2
$  element n°  prefa date     element n°  prefa date
     405         -12             406         -14

SUSPENDRE  CONCORDANCE $ necessary to enable the DEPLACES option
DATE  100
$ displacement of two elements cast in situ on the current date; their
$ extremity nodes are fixed (positive element na); the absence of
$ attributes
$ for the DEPLACES option requires that they have the same length and
$ orientation
PLACER  ELEMENTS  DEPLACES  2
$ element n°  element na     element n°  element na
     150         101             151         102

DEFINIR  STRUCTURE  POUSSEE  TAB_POU  ... $ also defines a rotation
ROTATION  ...
$ displacement of two elements by rotation, fixed origins; CALCUL
$ RHEOLOGIQUE FIN mode not required, because they are neither cast nor
$ prefabricated
PLACER  ELEMENTS  DEPLACES  ROTATION  TAB_POU  2
$ element n°  element na      element n°   element na
      60         -10              61         -11
```

Related commands

VERIFIER ; EDITER / NONEDITER ; DEFINIR STRUCTURE POUSSEE

EMPATTEMENT EQUIPAGES ; SUSPENDRE / CONTINUER CONCORDANCE

AFFECTER CONTRAINTES ; DATE ; PLACER EQUIPAGES

DEPLACER EQUIPAGES ; SUPPRIMER EQUIPAGES ; ACTIVER ELEMENTS

POUSSER [AUTOMATIQUEMENT] STRUCTURES

8.37 – PLACE ARTICULATIONS

PLACER ARTICULATIONS [CONTINUITE [$\left\{\begin{matrix}\text{ABSOLU}\\ \text{RELATIF}\end{matrix}\right\}$]] nb_artis

$$\left\langle \text{nom\_arti} \quad \text{no\_element} \quad \theta_1 \quad \theta_2 \quad \theta_3 \quad [\text{du}_x \quad \text{du}_y \quad \text{du}_z \quad \text{d}\theta_x \quad \text{d}\theta_y \quad \text{d}\theta_z] \right\rangle_{\text{nb\_artis}}$$

Parameters

- nb_artis: the number of articulations to be placed, positive.
- nom_arti: the name of an articulation-type, defined on an ARTICULATION command of the PH1 module (see Chapter 6).
- no_element: number of the element where it will be placed.
- θ_1, θ_2, θ_3: standardized triple rotation (see Figure 1.2) orienting the definition reference frame of the articulation-type, relative to the global reference frame (zero angles when these reference frames are the same).
- du_x, du_y, du_z, $d\theta_x$, $d\theta_y$, $d\theta_z$: displacement vector $\{dU_p\}$ in the global reference frame, to be used as a supplement of the differential vector of the preexisting displacements, to establish continuity (only provide if the CONTINUITE option is chosen). A positive displacement value du_i causes shortening, while a negative value causes the elongation of the articulation-type. A vector whose components are all zero establishes total continuity.

Functions

This command is used to activate a group of articulation elements, by assigning them articulation-types that can be reoriented relative to the global reference frame, and possibly predeformed.

Without the CONTINUITE option, the rigidity of the model is modified and no loadings are generated.

With the CONTINUITE option, the PH3 module totally (with the ABSOLU option and a null displacement vector $\{dU_p\}$) or partially establishes (in the other cases) the continuity of the accumulated displacements between the incident nodes and the articulations to be placed.

If the displacement vectors at the origin and end nodes of one of them are called $\{U_o\}$ and $\{U_e\}$, the differential vector of the displacements is:

$$\{dU_r\} = \{U_e\} - \{U_o\} \tag{8.1}$$

With the ABSOLU option (default option), the initial state $\{dU_r\}$(relative displacement vector) is not taken into account and the user directly sets the displacement vector $\{dU_a\}$ (absolute displacement vector) really used to establish the continuity.

$$\{dU_a\} = \{dU_p\}$$

With the RELATIF option, the initial state is taken into account and the user modifies this state by adding $\{dU_p\}$ to $\{dU_r\}$

$$\{dU_a\}=\{dU_r\}+\{dU_p\} \tag{8.2}$$

And the following operations are carried out:

- application of symmetrical loading to each correctly oriented articulation-type, intended to predeform it as per $\{dU_a\}$,
- modification of the rigidity of the model by introducing additional articulations,
- generation of the opposite predeformation loading,
- application of this loading to the active structure in its new configuration, and calculation of its state of deformation and solicitation.

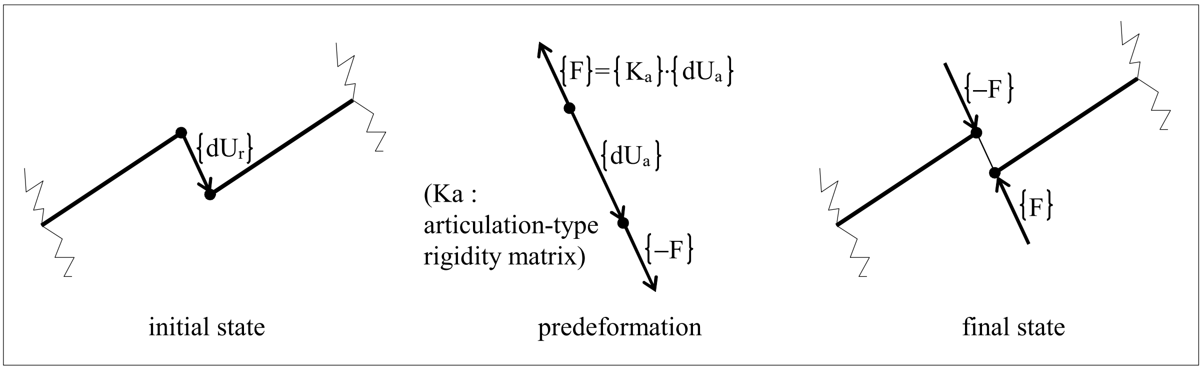


Figure 8.26 - Placement of an articulation with the CONTINUITE option

In DEPLACEMENTS NON REPERCUTES mode, when an articulation element has a free node (no links with a support or an active element) and a node linked to at least one deformed active element, its placement does not move the free node that has not yet been moved towards the node that has been. In DEPLACEMENTS REPERCUTES mode, the free node is moved.

In this way, when the support of a bridge deck on a "soft" pile is modeled by an articulation that is offset from the mean fiber of the deck using a rigid element, the displacements must be transferred to obtain the "right" deformation.

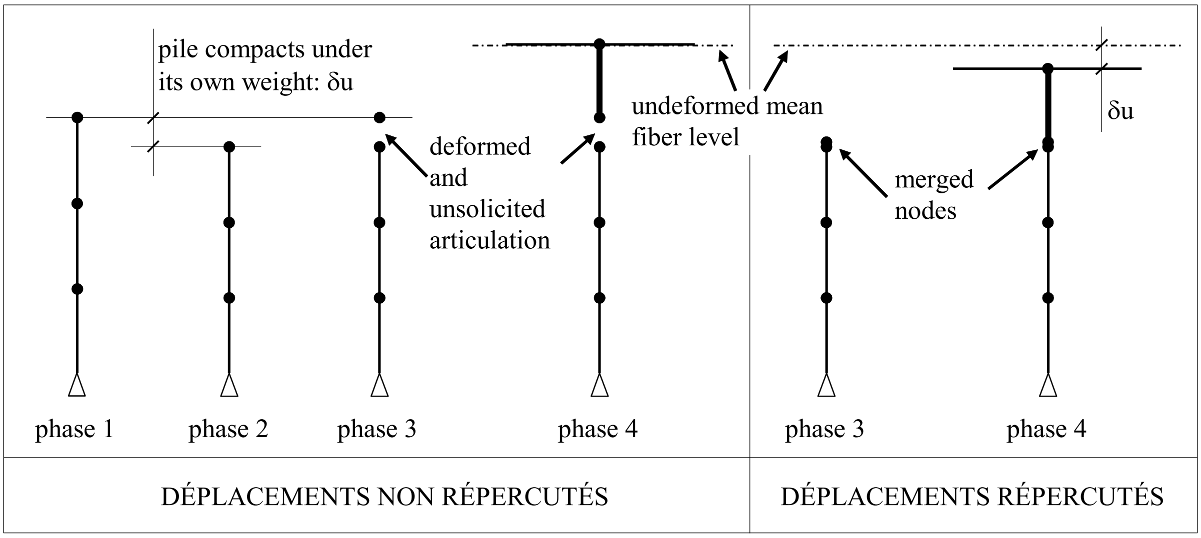


Figure 8.27 - Placement of an articulation, deformations transferred

Each unnumbered rigid element (defined or eccentricity) is automatically activated as soon as one of its incident nodes is linked to a placed articulation-type.

Conditions of use

- An pair of nodes cannot receive more than one articulation simultaneously.
- The same articulation-type can be assigned to several articulation elements.

Methodological advice

- The CONTINUITE option is often used to help simulate push operations.
- To simulate the jacking of an articulation between two nodes which have a different current differential displacement, use the option CONTINUITY with the ABSOLU option or the RELATIF option:
  - With the ABSOLU option, the jacking does not take into account the existing differential displacement and replace it by the new one setted by the user. If the later is null, both nodes have the same displacement.
  - With the RELATIF option, the differential displacement introduced by the user is algebraically added to the one that comes from the construction phasing.

Examples

```
$ three-span cantilever bridge; the links between the breaks
$ in the deck are modeled by two typical articulations called ARTI_1
REPERCUTER  DEPLACEMENTS
..............
$ activation of the side spans ansd the consoles of the central span
..............
PLACER  ARTICULATIONS  2
nom_arti  no_element  teta1  teta2  teta3
 ARTI_1      125        0.0    0.0    0.0
 ARTI_1      175        0.0    0.0    0.0
..............
$ activation of the rest of the central span
..............

$ placement of an articulation with total restoration of continuity
PLACER  ARTICULATIONS CONTINUITE ABSOLU 1
nom_arti  no_element  teta1 teta2 teta3  dux duy duz  dtetax dtetay
dtetaz
$                          (degrees)                          (radians)
 ARTI_C     12345       0.0    0.0   12.0   0.0 0.0 0.0    0.0    0.0    0.0
```

Related commands

VERIFIER ; EDITER / NONEDITER

REPERCUTER / NONREPERCUTER DEPLACEMENTS / TRANSLATIONS ; DATE

REMPLACER ARTICULATIONS ; ACTIVER ELEMENTS

SUPPRIMER ELEMENTS ; CAS DE CHARGE

POUSSER [AUTOMATIQUEMENT] STRUCTURES ; ETAT

8.38 – REPLACE ARTICULATIONS

REMPLACER ARTICULATIONS [CONTINUITE] nb_artis

$$\left\langle \text{nom\_arti} \quad \text{no\_element} \quad \theta_1 \quad \theta_2 \quad \theta_3 \quad [du_x \quad du_y \quad du_z \quad d\theta_x \quad d\theta_y \quad d\theta_z] \right\rangle_{\text{nb\_artis}}$$

Parameters

- nb_artis: the number of articulations to be replaced, positive.
- nom_arti: the name of a new articulation-type, defined on an ARTICULATION command of the PH1 module (see Chapter 6).
- no_element: number of the element where it will replace the old articulation-type.
- θ_1, θ_2, θ_3: standardized triple rotation (see Figure 1.2) orienting the definition reference frame of the new articulation-type, relative to the global reference frame (zero angles when these reference frames are the same).
- du_x, du_y, du_z, $d\theta_x$, $d\theta_y$, $d\theta_z$: see the PLACER ARTICULATIONS command.

Functions

This command is used to replace articulation-types assigned to a group of articulation elements, and to redefine their orientation.

Without the CONTINUITE option, the following operations are performed:

- modification of the rigidity of the model by removing the old articulations and introducing the new ones,
- generation of loading due to the removal of the old articulations, obtained by inverting the forces that they apply to the structure,
- application of this loading to the active structure in its new configuration, and calculation of its state of deformation and solicitation.

With the CONTINUITE option, the new articulations must be redefined before they are introduced, and the loading of the inverted predeformation must be superimposed on the loading due to the removal of the old articulations (analogy with the PLACER ARTICULATIONS command).

Methodological advice

☐ The CONTINUITE option is often used to help simulate push operations.

☐ To simulate the jacking of an articulation between two nodes which have a different current differential displacement, use the option CONTINUITY with the ABSOLU option or the RELATIF option:

- With the ABSOLU option, the jacking does not take into account the existing differential displacement and replace it by the new one setted by the user. If the later is null, both nodes have the same displacement.
- With the RELATIF option, the differential displacement introduced by the user is algebraically added to the one that comes from the construction phasing.

Examples

Consider the case of an articulation that is blocked then released. Two articulation-types have been defined: ARTI_B and ARTI_L, which respectively represent the blocked and released articulation.

```
$ installation of the blocked articulation, the temporary block
$ stabilizes the part of the structure that is being built
PLACER  ARTICULATIONS  1
ARTI_B  256  3*0.0
..............
$ the state of progress of the construction allows element 256
$ to be unblocked without compromising stability; ARTI_L replaces ARTI_B;
$ the loads previously supported by ARTI_B are applied to the new
structure
REMPLACER  ARTICULATIONS  1
$ nom_arti  no_element  teta1 teta2 teta3
 ARTI_L      256       0.0   0.0   0.0
```

Related commands

VERIFIER ; EDITER / NONEDITER ; DATE ; PLACER ARTICULATIONS ; ETAT

8.39 – ACTIVATE ELEMENTS

ACTIVER ELEMENTS $\left[{}^{*}\begin{Bmatrix}\text{POIDS}\\ \text{EINSTANTANE}\end{Bmatrix} \right]$ [nb_elements]

$[\langle \text{no\_element} \rangle_{\text{nb\_elements}}]$

Parameters

- nb_elements: the number of elements to be activated, positive.
- no_element: number of an element to be activated.

The numbers of the elements are only provided (in their actual order of activation, if it is meaningful) if nb_elements is also provided. Otherwise all the non-articulation elements of the model that are not yet activated will be activated simultaneously, by default.

Functions

For the module_reference, para_supp$_1$ and loi_module parameters, refer to the MATERIAU command of the GE1 module (see Chapter 3) or the PH1 module (see Chapter 6).

This command activates a group of non-articulation elements, with the optional integration of their dead weight.

Each unnumbered rigid or eccentricity element is automatically activated as soon as at least one of its incident nodes is linked to an element that is activated directly or indirectly by a rigid element. The supports do not bring about the activation of the rigid elements.

Without the POIDS option, the rigidity of the model is modified and no loadings are generated.

With the POIDS option, the following additional operations are performed:

- generation of loading due to the removal of the dead weight of the elements to be activated that are suspended from form travelers,
- generation of loading due to the dead weight of all the elements to be activated,
- application of these loadings to the active structure in its new configuration, and calculation of its state of deformation and solicitation.

Half of the dead weight of each biarticulated element is applied to its origin node and to its end node.

The calculated rigidity of the activated elements takes account of the properties of their constituent materials and of the selected calculation option for the rheological phenomena.

In CALCUL FORFAITAIRE mode, the Young's modulus of the constituent materials of the elements to be activated, used to calculate their rigidity, is:

- the reference modulus (module_reference parameter) by default,
- the para_supp$_1$ parameter, with the EINSTANTANE.option.

In CALCUL RHEOLOGIQUE FIN, the Young's modulus used is:

- the reference modulus (module_reference parameter) for materials whose loi_module parameter is zero;
- the para_supp$_1$ parameter (instantaneous modulus) for materials whose instantaneous modulus is constant over time,
- calculated according to the loi_module and para_supp$_1$ parameters and the casting date for elements with an instantaneous modulus that is variable over time.

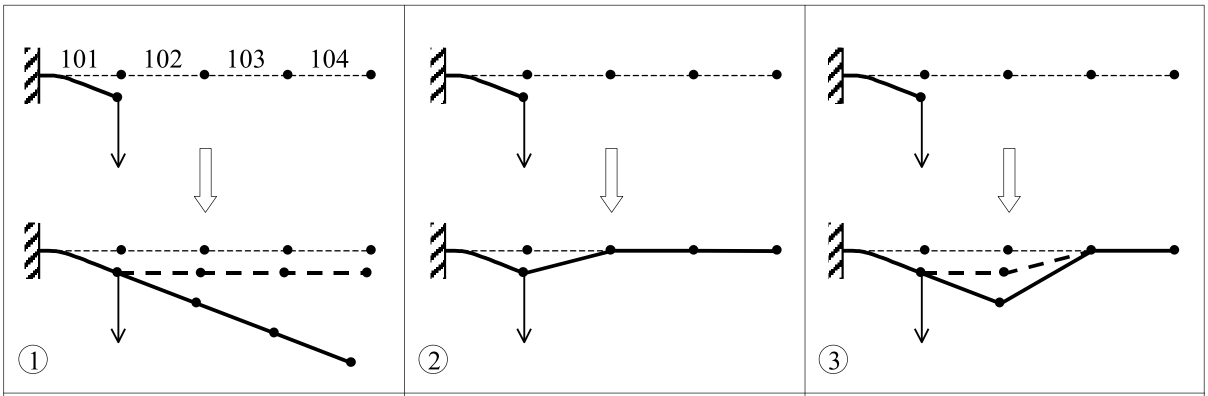
The displacements or translations caused by introducing new elements are controlled according to their transfer mode.

When a list of elements to be activated is provided, the transfer mode of the deformations influences the deformation produced.

In DEPLACEMENTS REPERCUTES mode, when an element has a free node (no links with a support or an active element) and a node linked to at least one deformed active element, its activation causes the transfer of the displacements (and rotations) of the linked node towards the free node.

The process is repeated in the prescribed order for all the elements to be activated.

For example, consider a four-element console whose first element is activated and loaded, and in which the three other elements are activated in the six possible different orders.



101 → 102 → 103 → 104	101 → 104 → 103 → 102 101 → 103 → 104 → 102 101 → 103 → 102 → 104	101 → 102 → 104 → 103 101 → 104 → 102 → 103

Figure 8.28 - Elements activated with DEPLACEMENTS / TRANSLATIONS REPERCUTES

In TRANSLATIONS REPERCUTEES mode, only pure displacements are affected, not rotations (see the bold dashed lines in Figure 8.20).

In DEPLACEMENTS NON REPERCUTES mode, all the previously free nodes of the activated elements remain on their "zero line" (non-deformed initial structure) and the order of activation is indifferent.

In the example above, the deformation of the case number ① is always obtained.

Conditions of use

- Activation by default of all the elements only works when no non-articulation elements are active.
- The articulation elements can only be activated by the PLACER ARTICULATIONS command.
- In CALCUL RHEOLOGIQUE FIN mode, all elements made of an aging material must be placed with the COULES or PREFABRIQUES option of the PLACER ELEMENTS command, before being activated.
- The date of implementation of a cast or prefabricated element must be before its date of activation.
- An element suspended from a form traveler must be activated with the POIDS option.
- Biarticulated elements activated with the POIDS option cannot be tensioned.

Examples

```
$ simultaneous activation of all non articulation elements
ACTIVER  ELEMENTS
```

```
$ activation of two elements with their specific weight
$ and their instantaneous Young’s modulus
ACTIVER  ELEMENTS  POIDS  EINSTANTANE  2
101  102
```

Related commands

VERIFIER ; CHAINETTES

REPERCUTER / NONREPERCUTER DEPLACEMENTS / TRANSLATIONS

EDITER / NONEDITER ; AFFECTER CONTRAINTES ; DATE

PLACER ARTICULATIONS ; PLACER ELEMENTS ; TENDRE ELEMENTS

SUPPRIMER ELEMENTS ; CAS DE CHARGE

POUSSER [AUTOMATIQUEMENT] STRUCTURES

8.40 – TENSION ELEMENTS

$$\text{TENDRE} \quad \text{ELEMENTS} \quad \left[\left\{\begin{matrix}\underline{\text{RELATIF}}\\ \text{ABSOLU}\end{matrix}\right\}\right] \quad \left[\left\{\begin{matrix}\mathbf{FX1}\\ \mathbf{FY1}\\ \mathbf{FZ1}\\ \mathbf{FX2}\\ \mathbf{FY2}\\ \mathbf{FZ2}\\ \underline{\mathbf{FC1}}\\ \mathbf{FC2}\\ \mathbf{FT1}\\ \mathbf{FT2}\\ \mathbf{LGV}\end{matrix}\right\}\right] \quad \text{nb\_elements}$$

$$\left\langle \text{no\_element} \quad \text{tension\_l} \right\rangle_{\text{nb\_elements}}$$

Parameters

- nb_elements: number of biarticulated elements to be tensioned, positive.
- no_element: number of an element.
- tension_l: variation of length when unladen (RELATIF mode) or length when unladen (ABSOLU mode) imposed if the LGV option is used. Otherwise, variation of tension (RELATIF mode) or tension (ABSOLU mode) to be applied to a point and according to an axis defined by the options FX1 to FT2 (positive for traction according to chord or tangent).

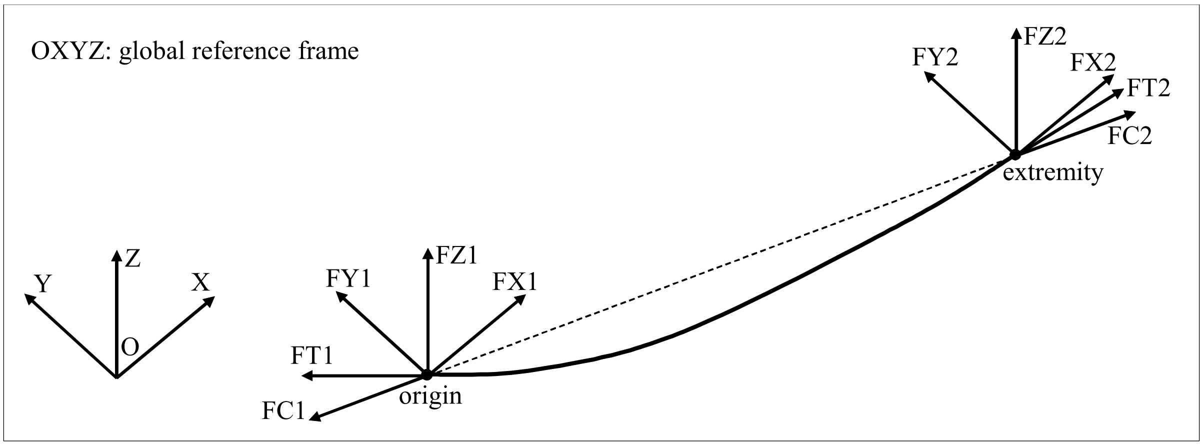


Figure 8.29 - Options for the application of tensions to biarticulated elements

Functions

This command is used to fix or modify the unladen length of a group of biarticulated elements, or to apply certain variations of tension to their origins or extremities, according to an axis of the global reference frame, axes tangential to their route or according to their chords.

With the RELATIF option (selected by default), tension_l is added to the unladen length or corresponding tension, in the no_element element.

With the ABSOLU option, tension_l is the unladen length or the tension that is imposed on it.

With the options FX1, FY1, FZ1, FX2, FY2 or FZ2, tension_l is a tension applied in the OX, OY or OZ axis of the global reference frame, to the origin or the extremity of the no_element element.

With the options FC1 or FC2, tension_l is a tension applied according to its chord, to its origin (default option) or to its extremity. With the options FT1 or FT2, tension_l is a tension applied according to the tangent, to its origin or to its extremity.

Since the element is fictively cut near the node in question, the tension applied is always the action of its extremity part on its origin part.

With the LGV option, tension_l is an unladen length.

Between their activation and their tensioning, the elements declared as chains (see CHAINETTES command) are assumed to be infinitely supple.

The tension of elements that are not declared as of the chain type is applied without taking their dead weight into consideration, whereas the dead weight of the elements declared to be of the chain type is still applied by the PH3 module, when they are tensioned for the first time.

The tension fixed in ABSOLU mode in the elements covered by the simplified chains mode is a mean tension, irrespective of its point of application, because their dead weight is applied by the PH3 module, half to their origins and half to their extremities.

Conditions of use

- Only active biarticulated elements can be tensioned.
- Biarticulated elements activated with the POIDS option of the ACTIVER ELEMENTS command cannot be tensioned.
- The elements covered by the chains calculation mode must always be tensioned for the first time in ABSOLU mode.

Examples

```
$ declaration of the chain type elements (chain calculation mode)
CHAINETTES  2
100  0.50    101  0.50
..............
ACTIVER  ELEMENTS  2
101  102
$ first tensioning according to their cords, always in ABSOLU mode,
$ option FC1 is applied by default; the elements contribute to the
$ rigidity of the model; their own weight is jointly applied
TENDRE  ELEMENTS  ABSOLU  2
100  160.0    101  150.0
..............
$ retensioning according to their cords; options RELATIF and FC1 are
$ applied
$ by default; the values introduced are added to the current tensions
TENDRE  ELEMENTS  2
100  0.10    101  0.10
```

This script would be the same to tension elements that are not declared as being of the chain type.

Related commands

VERIFIER ; UNITES ; CHAINETTES ; EDITER / NONEDITER ; DATE

ACTIVER ELEMENTS ; CAS DE CHARGE ; REDISTRIBUER ; ETAT

8.41 – REMOVE ELEMENTS

SUPPRIMER ELEMENTS nb_elements

$\langle \text{no\_element} \rangle_{\text{nb\_elements}}$

Parameters

- nb_elements: number of elements to be removed, positive.
- no_element: number of an element to be removed.

Functions

This command is used to deactivate a group of active elements of all types by performing the following operations:

- modification of the rigidity of the model by removing the selected elements,
- generation of loading due to this removal, obtained by inverting the forces that they apply to the structure,
- application of this loading to the active structure in its new configuration, and calculation of its state of deformation and solicitation.

If necessary, the loading history of the removed elements is disabled.

Conditions of use

- ❐ This command can only be introduced is the structure has at least one placed support and one active element.
- ❐ The stability of the structure must be maintained after the removal of the elements.
- ❐ All active elements can be removed, including articulations.
- ❐ A removed element can be placed or activated again.

Examples

```
  ACTIVER  ELEMENTS  6
105  110  505  510  705  710
  PLACER  ARTICULATIONS  3
  ARTI_1  1001  3*0.0
  ARTI_1  1005  3*0.0
  ARTI_1  1007  3*0.0
  ..............
  SUPPRIMER ELEMENTS  3
  505  510  1005        $ the removed elements include an articulation
```

Related commands

VERIFIER ; EDITER / NONEDITER ; DATE ; PLACER APPUIS

REMPLACER APPUIS ; SUPPRIMER APPUIS ; PLACER ARTICULATIONS

ACTIVER ELEMENTS ; CAS DE CHARGE

POUSSER [AUTOMATIQUEMENT] STRUCTURES ; ETAT

8.42 – TENSION CABLES

TENDRE CABLES [NONINJECTES] [nb_cables]

$[\langle \text{nom\_cable} \rangle_{\text{nb\_cables}}]$

Parameters

- nb_cables: number of cables to be tensioned, positve.
- nom_cable: name of a cable to be tensioned, defined on a TRACE CABLE command of the GE1 module (see Chapter 3), or generated by the MC1 module (see Chapter 4), or PH1 module (see Chapter 6).

The names of the cables are only provided if nb_cables is also provided. Otherwise, all the cables of the model will be tensioned simultaneously, by default.

Functions

This command causes a group of cables to be tensioned, with or without injection of their sheaths, by performing the following operations:

- generation of loading due to their initial tensions,
- application of this loading to the active structure and calculation of its new state of deformation and solicitation,
- generation of loading due to their instantaneous losses of tension (which include elastic losses in the CALCUL FORFAITAIRE mode) and update of their tensions,
- application of this loading to the active structure and calculation of its new state of deformation and solicitation.

Without the NONINJECTES options, the cables are assumed to be linked to the structure along their entire length (sheaths injected in full, by default).

With the NONINJECTES option, the cables are assumed to be linked to the structure only by their anchor points. This option is used:

- by the PH3 module, in CALCUL RHEOLOGIQUE FIN mode, to take account of interactions between the concrete and the steels,
- by the ENV module, in all cases, to calculate the stress on the basis of the forces due to the operating loads.

Conditions of use

- ❐ The tensioning of all the cables by default only works when none of the cables are tensioned.
- ❐ Only cables that have not yet been tensioned can be tensioned.
- ❐ All the beam elements that are partially or totally crossed by tensioned cables must be active.

Methodological advice

- The NONINJECTES option must be used for cables or parts of cables inside the concrete, but that are not linked to their sheaths by injected slurry, or for cables outside the concrete (normally modeled with a zero sheath diameter in the corresponding CARACTERISTIQUES CABLES commands of the GE1 module, see Chapter 3).
- To obtain cables injected in sections, they must be tensioned with the NONINJECTES option, then injected, cable by cable, using the PARTIELLEMENT option of the INJECTER CABLE[S] command.

Examples

```
$ tensioning of all the cables of the model
$ with injection of their sheaths
TENDRE  CABLES

$ tensioning of a couple of cables, without injection of their sheaths
TENDRE  CABLES  NONINJECTES  2
P1F119G  P1F119D
```

Related commands

VERIFIER ; EDITER / NONEDITER ; AFFECTER CONTRAINTES

DATE ; INJECTER CABLE[S] ; DETENDRE CABLES ; PERTES CABLES

IMPRIMER TENSIONS

8.43 – INJECT CABLE[S]

$$\text{INJECTER}\quad\left\{\begin{array}{l}\text{CABLES}\quad[\text{nb\_cables}]\\ \text{CABLE}\quad\text{PARTIELLEM ENT}\quad\text{nom\_cable\_p}\quad\text{nb\_element s}\end{array}\right\}$$

$$\left\{\begin{array}{l}[\langle\text{nom\_cable\_t}\rangle_{\text{nb\_cables}}]\\ \langle\text{no\_element}\rangle_{\text{nb\_elements}}\end{array}\right\}$$

Parameters

- nb_cables: the number of cables to be injected, in the event of total injection of a group of cables, positive. If it is not provided, all the tensioned cables will be injected.
- nom_cable_t: name of a cable to be totally injected. The list of names is only provided if nb_cables is also provided.
- nom_cable_p: name of the cable to be partially injected.
- nb_elements: number of crossed elements that will be injected, positive.
- no_element: number of a crossed element. The list of numbers is only provided if the PARTIELLEMENT option is used.

Functions

This command causes the injection of a group of cables along their entire length, or of a cable along a length reduced to a subset of the elements that it crosses.

Conditions of use

- ☐ The designated cables must be tensioned and not injected in the elements affected by the injection.
- ☐ Injection is an irreversible operation.

Methodological advice

See also the methodological advice for the TENDRE CABLES command.

Examples

```
$ injection of all the taut cables along their entire length
INJECTER  CABLES
```

```
$ injection of two cables through three elements
INJECTER  CABLE  PARTIELLEMENT  C2P10G  3
101  110  120
INJECTER  CABLE  PARTIELLEMENT  C2P10D  3
101  110  120
```

Related commands

DATE ; TENDRE CABLES ; DETENDRE CABLES

8.44 – LOOSEN CABLES

DETENDRE CABLES [nb_cables]

$[\langle \text{nom\_cable} \rangle_{\text{nb\_cables}}]$

Parameters

- nb_cables: number of cables to be loosened, positive.
- nom_cable: name of a cable to be loosened.

The names of the cables are only provided if nb_cables is also provided. Otherwise, all the tensioned cables will be loosened simultaneously, by default.

Functions

This command is used to loosen a group of cables by performing the following operations:

- generation of loading due to their current losses of tension (accumulated since they were tensioned),
- application of this loading to the active structure and calculation of its new state of deformation and solicitation,
- generation of loading due to the removal their initial tensions,
- application of this loading to the active structure and calculation of its new state of deformation and solicitation.

Conditions of use

☐ The designated cables must be tensioned.

☐ A loosened cable can be tensioned again.

Examples

```
$ simultaneous loosening of all the taut cables
DETENDRE  CABLES
```

```
$ loosening of a couple of cables
DETENDRE  CABLES  2
P1F119G  P1F119D
```

Related commands

VERIFIER ; EDITER / NONEDITER ; DATE ; TENDRE CABLES

INJECTER CABLES ; PERTES CABLES ; IMPRIMER TENSIONS

8.45 – CABLE LOSSES

PERTES CABLES [RATIO pourcent] [nb_cables]

$[\langle \text{nom\_cable} \rangle_{\text{nb\_cables}}]$

Parameters

- pourcent: percentage of the total deferred losses of prestress to be introduced. Must be greater than 1.0 and less than or equal to 100.0. The default value is 100.0.
- nb_cables: the number of selected cables, positive. If not provided, all the tensioned cables will be taken into consideration by default.
- nom_cable: name of a cable. The list of names is only provided if nb_cables is also provided.

Functions

This command is used to introduce a fraction of the total deferred losses of tension of a group of tensioned cables, by performing the following operations:

- generation of loading due to the specified percentage of losses and update of the tensions in the corresponding cables,
- application of this loading to the active structure and calculation of its new state of deformation and solicitation.

Conditions of use

- ☐ This command can only be used in CALCUL FORFAITAIRE mode.
- ☐ The losses of a group of cables can be introduced in several steps using ratios that always amount to their total values.
- ☐ No check is made that the sum total of the percentages applied to a group of cables does not exceed 100.0.

Examples

```
$ choice of the FORFAITAIRE calculation mode
AFFECTER  CONTRAINTES  UNIQUES  500.0
..............
$ application of the total deferred losses of tension of the taut cables
PERTES  CABLES
..............
$ application of one half of the deferred losses of tension of two cables
PERTES  CABLES  RATIO  50.0  2
F1_C1G  F1_C1D
..............
$ application of one quarter of the deferred losses of tension
PERTES  CABLES  RATIO  25.0  2
F1_C1G  F1_C1D
$ at this point, one quarter of their total losses remain to be applied
...
```

Related commands

VERIFIER ; EDITER / NONEDITER ; AFFECTER CONTRAINTES ; DATE

TENDRE CABLES ; DETENDRE CABLES ; IMPRIMER TENSIONS

8.46 – LOAD CASE

```
CAS   DE   CHARGE   [no_charge]
     titre_charge
    |...............................|
    |commandes de chargement|
    |...............................|
```

Parameters

- no_charge: identification number of the load case to be created, positive and less than 90_000. Is only provided if the effects of the load case must be recorded in the database.
- titre_charge: title indicating its content and designed to facilitate its subsequent control (character string).

The immediately following loading commands give the composition of the load case. It may be a DEFORMATION APPUIS command, which must be processed individually, or any combination of any number of the commands:

- CHARGEMENT [IDENTIQUE] NOEUDS ;
- CHARGEMENT POUTRE / [IDENTIQUE] ELEMENTS ;
- DISTORSION ELEMENTS ;
- CHARGEMENT [IDENTIQUE] THERMIQUE.

Functions

This command starts a load case, identifies it and manages the recording of its effects in the database.

After analyzing the corresponding loading commands, the following operations are performed:

- combination of all the loads,
- application to the active structure and calculation of their effects on the deformations and solicitations,
- in CUMUL mode only, calculation of the new state of deformation and solicitation of the active structure,
- if an identification number is provided, recording in the database of all the effects produced in the active structure on the displacements, support reactions, forces and possible stress (the list of placed supports and active elements is added).

A recorded load case is identified by its number and, possibly, by the name of the last saved structure using a SAUVER command, or is restored using the SUITE option of the PHASES command.

It can be called (with its title) by the ETU module, with a view to weighting its effects, integrating them in a combination or an envelope or editing them, or by the RES module, to view its effects.

Conditions of use

- The identification number of an EFFET already recorded by the PH3, ENV, DYN or ETU modules (according to the terminology used by the ETU module, see Table 12.1) cannot be used to designate a recorded CAS DE CHARGE.
- A recorded load case can be replaced in the database by another one with the same number. This possibility must only be used to update the effects of an erroneous load case.
- Attaching several load cases to a saved structure is purely formal and can correspond to different static situations.
- Only the active elements and the nodes demarcating them can be loaded.
- Undeclared biarticulated elements of the chain type (see CHAINETTES command) can only be loaded axially, for example to introduce their tensions (concurrently with the TENDRE ELEMENTS command). Remember that their dead weight must be introduced in the form of loads at the nodes.
- The declared biarticulated elements of the chain type can only be subjected to axial thermal loadings. Their tensioning must be controlled by the TENDRE ELEMENTS command.

Examples

```
PHASES $ session 1
..............
CAS DE CHARGE  100
'TASSEMENT D''APPUIS'
$ this loading command cannot be associated with any others
DEFORMATION  APPUIS  ...
..............
SAUVER  STRUCT1
CAS DE CHARGE  101
'ANCIENNE DEFINITION'
$ these commands can be associated without restriction
CHARGEMENT  NOEUDS  ...
..............
CHARGEMENT  ELEMENTS  ...
..............
DISTORSION  ELEMENTS  ...
..............
CHARGEMENT  THERMIQUE  ...
..............
FIN

PHASES  SUITE  STRUCT1 $ session 2
..............
CAS DE CHARGE  101
'NOUVELLE DEFINITION'
..............
FIN
```

At the end of the analysis of these two examples, the database contains:

- the CAS DE CHARGE 100 (not attached to a saved structure),
- the new definition of CAS DE CHARGE 101, attached to the saved structure STRUCT1; can be called by the ETU module by the name: « CHARGE 101   STRUCTURE STRUCT1 ».

Related commands

PHASES ; VERIFIER ; CHAINETTES ; SUSPENDRE / CONTINUER CUMUL

EDITER / NONEDITER ; DATE ; PLACER ARTICULATIONS

ACTIVER ELEMENTS ; TENDRE ELEMENTS ; SUPPRIMER ELEMENTS

CHARGEMENT [IDENTIQUE] NOEUDS / EXTREMITES

DEFORMATION APPUIS ; CHARGEMENT [POUTRE] / [IDENTIQUE] ELEMENTS

DISTORSION ELEMENTS ; CHARGEMENT [IDENTIQUE] THERMIQUE

SAUVER ; ETAT

8.47 – [IDENTICAL] LOADING NODES / EXTREMITIES

$$\text{CHARGEMENT} \quad [\text{IDENTIQUE}] \quad \left\{ \begin{matrix} \text{NOEUDS} \\ \text{EXTREMITES} \end{matrix} \right\} \quad \text{nb\_noel}$$

$$\left\{ \begin{matrix} \left\langle \left\{ \begin{matrix} \text{no\_noeud} \\ [\left\{ \begin{matrix} + \\ - \end{matrix} \right\}]\text{no\_element} \end{matrix} \right\} \quad Fx \quad Fy \quad Fz \quad Mx \quad My \quad Mz \right\rangle_{\text{nb\_noel}} \\ \left\langle \left\{ \begin{matrix} \text{no\_noeud} \\ [\left\{ \begin{matrix} + \\ - \end{matrix} \right\}]\text{no\_element} \end{matrix} \right\} \right\rangle_{\text{nb\_noel}} \quad Fx \quad Fy \quad Fz \quad Mx \quad My \quad Mz \end{matrix} \right\}$$

Parameters

- nb_noel: number of nodes or "extremities" of elements to be loaded, positive.
- no_noeud: number of a node to be loaded, to be provided if the NOEUDS option is used.
- no_element: number of an element, preceded by the *minus* sign if its origin is designated, or by the *plus* sign, or no sign at all, if its extremity is designated. To be provided if the EXTREMITES option is used.
- Fx, Fy, Fz, Mx, My, Mz: forces and moments applied in the OX, OY and OZ axes of the global reference frame, to the node no_noeud if the IDENTIQUE option is not used, or to the group of nodes whose numbers precede it, if the IDENTIQUE option is used.

Functions

This command is used to load, in the global reference frame, individually or collectively, a group of numbered nodes or a group of invisible nodes, linked to other nodes by rigid eccentricity elements, and designated implicitly by their adjacent elements.

Examples

```
CAS DE CHARGE
’2 NOEUDS CHARGES INDIVIDUELLEMENT, 10 NOEUDS ET EXTREMITES
COLLECTIVEMENT’
CHARGEMENT  NOEUDS  2
$  node n°   Fx    Fy    Fz    Mx    My    Mz
     105      0.0   0.0 -25.0   0.0   0.0   0.0
     205      0.0   0.0 -12.5   0.0   0.0   0.0
CHARGEMENT  IDENTIQUE  NOEUDS  10
105 A 150 INC 5  $ node numbers
$  Fx    Fy    Fz    Mx    My    Mz
  10.0   0.0  25.0   0.0 300.0   0.0
CHARGEMENT  IDENTIQUE  EXTREMITES  10
 -5 A -25 INC -5  $ numbers of elements with loaded origins
105 A 125 INC  5  $ numbers of elements with loaded extremities
$  Fx    Fy    Fz    Mx    My    Mz
  10.0   0.0  25.0   0.0 300.0   0.0
```

Related commands

UNITES ; CHAINETTES ; CAS DE CHARGE

8.48 – SUPPORT DEFORMATION

DEFORMATION APPUIS nb_appuis

$$\left\langle \text{no\_noeud} \quad du_x \quad du_y \quad du_z \quad d\theta_x \quad d\theta_y \quad d\theta_z \right\rangle_{\text{nb\_appuis}}$$

Parameters

- nb_appuis: the number of supports to be deformed, positive.
- no_noeud: number of a support node.
- du_x, du_y, du_z, $d\theta_x$, $d\theta_y$, $d\theta_z$: displacements and rotations (in radians) imposed on its assigned support-type (relative to its state of deformation), according to the Ax_a, Ay_a and Az_a axes of its local reference frame (see Figure 8.13). A zero value designates a non-imposed deformation (and not a zero imposed deformation).

Functions

This command is used to impose variations in deformations on certain types of support-types in place, according to the chosen components of their specific reference frames.

No checks are made that each support-type in question has a stiffness in the directions of all the imposed displacements and rotations. When this is not the case (the coefficient of rigidity is zero), the support reaction component that appears is the action to be applied to the adjacent node in order to move or turn it by the required quantity.

Conditions of use

☐ A perfect or offset support can be deformed.

Examples

```
PLACER  APPUIS  2
APP1  1050  0.0 0.0 0.0   -1.0  30.0  0.0
APP1  2050  0.0 0.0 0.0   -1.0  25.0  0.0
..............
$ this load case can only include one command of this type
CAS DE CHARGE
'TASSEMENT DIFFERENTIEL DE DEUX APPUIS'
$ the supports are deformed in the global coordinates, only per Uz,
$ since the Az axes of their specific coordinates are merged
$ with the global OZ, while the other deformations remain free
DEFORMATION  APPUIS  2
$   node n°    dux   duy   duz      dtetax   dtetay   dtetaz
      1050      0.0   0.0  -0.05      0.0      0.0      0.0
      2050      0.0   0.0  -0.05      0.0      0.0      0.0
```

Related commands

PLACER APPUIS ; REMPLACER APPUIS ; SUPPRIMER APPUIS

CAS DE CHARGE

8.49 - [IDENTICAL] LOADING BEAM / ELEMENTS

$$\text{CHARGEMENT} \begin{Bmatrix} \text{POUTRE} \\ [\text{IDENTIQUE}]\ \text{ELEMENTS} \end{Bmatrix} \begin{Bmatrix} \text{CONCENTRE} \\ \text{REPARTI} \end{Bmatrix} \left(\begin{Bmatrix} \text{GLOBAL} \\ \underline{\text{LOCAL}} \\ \text{ARBITRAIRE} \end{Bmatrix} \right)$$

$$\begin{Bmatrix} \text{nop} \\ \text{nbe} \end{Bmatrix}$$

$$\begin{Bmatrix} \text{xr}_1 & [\text{xr}_2] & \text{no\_comp} & \text{q}_1 & [\text{q}_2] & [\theta_1 & \theta_2 & \theta_3] \\ \langle \text{no\_element} & \text{xr}_1 & [\text{xr}_2] & \text{no\_comp} & \text{q}_1 & [\text{q}_2] & [\theta_1 & \theta_2 & \theta_3] \rangle_{\text{nbe}} \\ \langle \text{no\_element} \rangle_{\text{nbe}} & \text{xr}_1 & [\text{xr}_2] & \text{no\_comp} & \text{q}_1 & [\text{q}_2] & [\theta_1 & \theta_2 & \theta_3] \end{Bmatrix}$$

Parameters

- nop: number of the beam to be loaded, to be provided if the POUTRE option is used.
- nbe, no_element: number of elements to be loaded, positive, and number of an element to be loaded. To be provided if the ELEMENTS option is used.

The load described below applies to all the active elements of the beam nop, to the element no_element, if the IDENTIQUE option is not used, or to the group of elements whose numbers precede it, if the IDENTIQUE option is used.

- xr_1: relative abscissa of the point of application of the concentrated load (CONCENTRE option) or of the start of the distributed load (linearly, REPARTI option).
- xr_2: relative abscissa of the point of application of the end of the distributed load, to be provided if the REPARTI option is used.

xr_1 and xr_2 are values between 0.0 and 1.0 (including the boundaries), expressed according to the ox axis of the local reference frame of each loaded element. xr_2 must be greater than xr_1.

- no_comp: number of the component of force or moment to be applied (1 Fx, 2 Fy, 3 Fz, 4 Mx, 5 My, 6 Mz) according to an axis of the GLOBAL reference frame, of the LOCAL reference frame of each loaded element (default value), or of an ARBITRAIRE reference frame (depending on the chosen option).
- q_1: intensity of the concentrated load, or at the start of the distributed load.
- q_2: intensity at the end of the distributed load, to be provided if the REPARTI option is used.
- θ_1, θ_2, θ_3: standardized triple rotation (see Figure 1.2) orienting the arbitrary loading reference frame, relative to the global reference frame (zero angles when these reference frames are the same), to be provided when the ARBITRAIRE option is used.

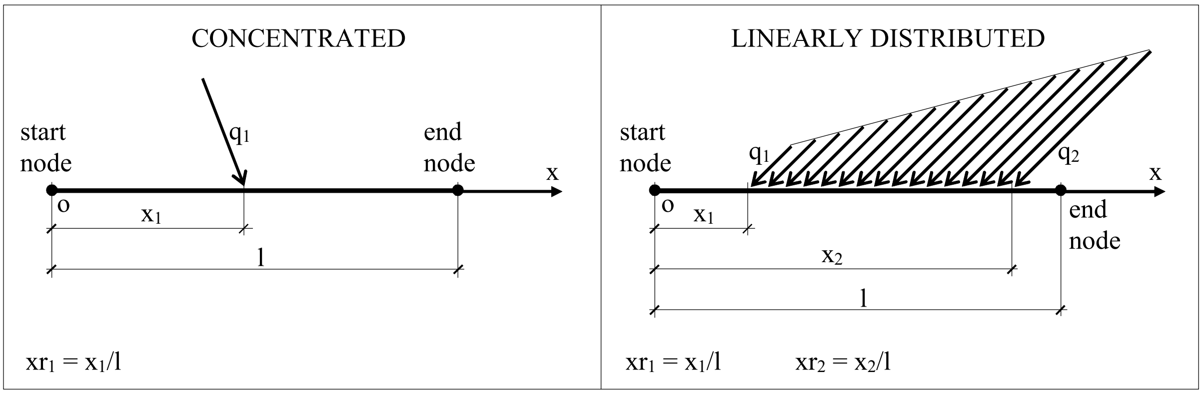


Figure 8.30 - Loads applicable to an element

Functions

This command is used to load a group of elements, which may belong to a beam, in a concentrated or linearly distributed manner.

The load is applied individually or collectively, according to an axis of the global reference frame, of the local reference frame of each loaded element, or of a unique and arbitrary reference frame.

Examples

```
CAS DE CHARGE
'CHARGEMENT  SUR ELEMENTS'
$ linearly distributed moment that is applied totally in the Y axis of an
$ arbitrary coordinate system
$ to all the active elements of a beam
CHARGEMENT  POUTRE  REPARTI  ARBITRAIRE  51
$  xr1   xr2     no_comp     q1     q2     teta1    teta2    teta3
   0.0   1.0         5      -50.0  -50.0     0.0     25.0     30.0
$ linearly distributed force partially applied in the axis OZ
$ of the global coordinate system, on a group of elements
CHARGEMENT  IDENTIQUE  ELEMENTS  REPARTI  GLOBAL  8
101 A 108
$  xr1   xr2     no_comp     q1     q2
   0.0   0.6         3     -100.0 -150.0
$ concentrated forces applied in axes ox and oy of the local coordinates,
$ to the middle of two elements
CHARGEMENT  ELEMENTS  CONCENTRE  LOCAL  2
$  element no  xr1   no_comp     q1
      101        0.5      1      100.0
      102        0.5      2      200.0
```

Related commands

UNITES ; CAS DE CHARGE

8.50 – LOADING BEAM / FIBRE_REPERE ELEMENTS

$$\text{CHARGEMENT} \begin{Bmatrix} \text{POUTRE} \\ \text{ELEMENTS} \end{Bmatrix} \begin{Bmatrix} \text{CONCENTRE} \\ \text{REPARTI} \end{Bmatrix} \text{FIBRE\_REPERE} \begin{Bmatrix} \text{nop} \\ \text{nbe} \end{Bmatrix}$$

$$\begin{Bmatrix} xc_1 \ \ [xc_2] \ \ \text{no\_comp} \ \ q_1 \ \ [q_2] \\ \langle \text{no\_element} \ \ xc_1 \ \ [xc_2] \ \ \text{no\_comp} \ \ q_1 \ \ [q_2] \ \rangle_{\text{nbe}} \end{Bmatrix}$$

Parameters

- nop: number of the beam to be loaded, to be provided if the POUTRE option is used.
- nbe, no_element: number of elements to be loaded, positive, and number of an element to be loaded. To be provided if the ELEMENTS option is used.

The load described below applies relatively to the beam nop or to the beam element no_element.

- xc_1: absolute curvilinear abscissa of the point of application of the concentrated load (CONCENTRE option) or of the start of the distributed load (linearly, REPARTI option), measured from the origin of the POUTRE nop or from the origin of the element no_element otherwise.
- xc_2: curvilinear abscissa of the point of application of the end of the distributed load, to be provided if the REPARTI option is used.

xc_1 and xc_2 are values of curvilinear abscissas expressed according to the ox axis of the reference fiber of the loaded POUTRE or of the POUTRE of the loaded element. They must be such that all the loaded elements are ACTIFS. In the case of an explicit loading of the POUTRE, the origin of the abscissas is the first point of the beam. In the case of an explicit loading of a beam element, the origin of the abscissas is the point of the reference fiber of the beam that precedes the element in question.

- no_comp: number of the force or moment component to be applied (1 Fx, 2 Fy, 3 Fz, 4 Mx, 5 My, 6 Mz) according to one of the axes of the reference fiber of the point of origin of each loaded element.

Note: The point of application of the load is assumed to move along the X axis of the reference fiber. Moments of torsion must, therefore, be calculated and applied relative to this axis.

- q_1: intensity of the concentrated load, or at the start of the distributed load.
- q_2: intensity at the end of the distributed load, to be provided if the REPARTI option is used.

Functions

This command is used to load a group of elements belonging to a beam, in a concentrated or linearly distributed manner, by providing the curvilinear abscissa along the reference fiber of the beam in question as the point of application.

Examples

```
CAS DE CHARGE
'CHARGE DE TROTTOIR SUR PONT COURBE'
$ 50 MN pavement offset by 3m along a length of 25m
CHARGEMENT  POUTRE  REPARTI  FIBRE_REPERE  51
$  xc1   xc2     no_comp     q1     q2
   0.0   25.0       3      -50.0  -50.0
CHARGEMENT  POUTRE  REPARTI  FIBRE_REPERE  51
$  xc1   xc2     no_comp     q1     q2
   0.0   25.0       4      -150.0  -150.0

$ concentrated forces applied in axis oZ of the fiber,
$ to 0.50 on either side of the point of origin of element 101
CHARGEMENT  ELEMENTS  CONCENTRE  FIBRE_REPERE  2
$  no_element   xc1   no_comp     q1
      101       -0.5      3      100.0
      101       +0.5      3      200.0
```

Related commands

UNITES ; CAS DE CHARGE,FIBRE REPERE

8.51 – ELEMENT DISTORSION

$$\text{DISTORSION} \quad \text{ELEMENTS} \quad \begin{Bmatrix} \text{CONCENTREE} \\ \text{REPARTIE} \end{Bmatrix} \quad \text{nb\_elements}$$

$$\left\langle \text{no\_element} \quad xr_1 \quad [xr_2] \quad \text{no\_comp} \quad q_1 \quad [q_2] \right\rangle_{\text{nb\_elements}}$$

Parameters

- nb_elements: number of elements to be loaded, positive.
- no_element: number of an element to be loaded.
- xr_1: relative abscissa of the point of application of the concentrated distortion (CONCENTREE option) or of the start of the distributed distortion (linearly, REPARTIE option).
- xr_2: relative abscissa of the point of application of the end of the distributed distortion, to be provided if the REPARTIE option is used.

xr_1 and xr_2 are values between 0.0 and 1.0 (including the boundaries), expressed according to the x axis of the local reference frame of each loaded element. xr_2 must be greater than xr_1.

- no_comp: number of the displacement or rotation component to be applied (1 u_x, 2 u_y, 3 u_z, 4 θ_x, 5 θ_y, 6 θ_z) according to an axis of the local reference frame of each loaded element.
- q1: intensity of the concentrated distortion, or at the start of the distributed distortion.
- q2: intensity at the end of the distributed distortion, to be provided if the REPARTIE option is used.

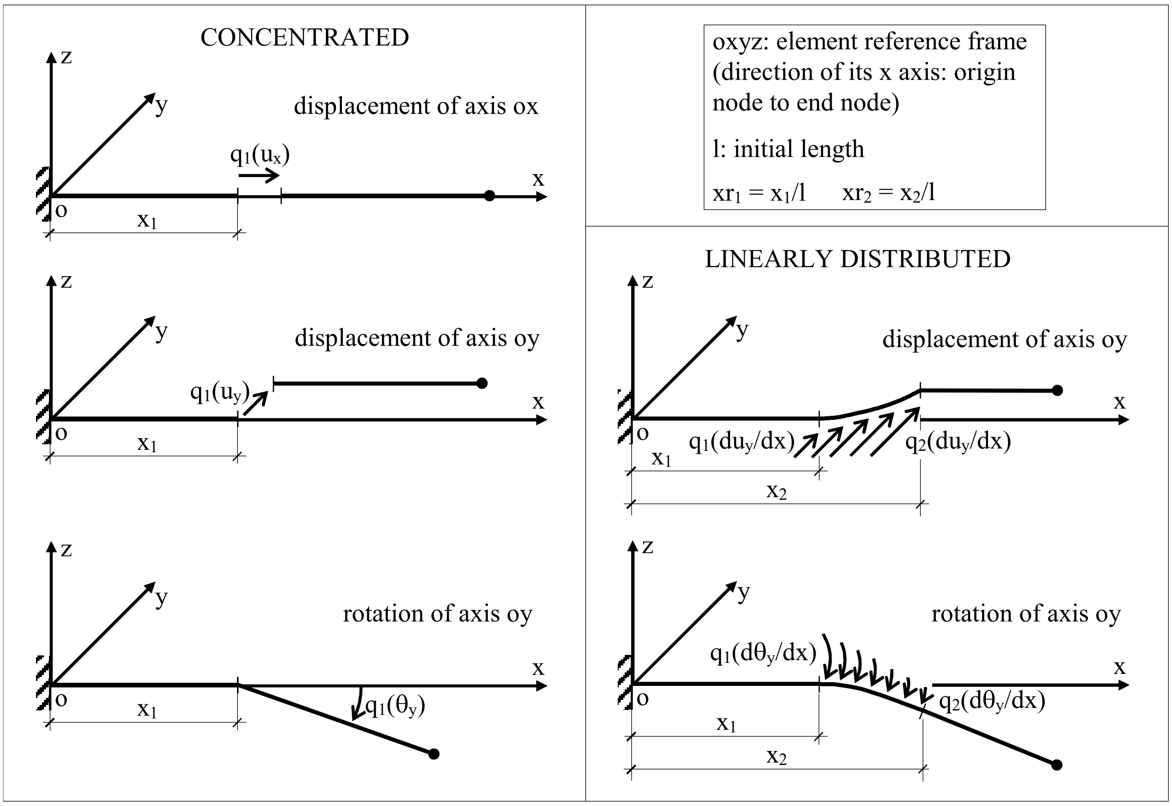


Figure 8.31 - Examples of distortions of elements

A distortion of an element is equivalent to a loading to be applied to it in order to deform it in the same manner, when it is assumed to be free.

Concentrated displacements are expressed in meters, and concentrated rotations in radians. Distributed displacements have no unit and distributed rotations are expressed in radian/meters.

Functions

This command is used to distort a group of elements in a concentrated or linearly distributed manner.

The load equivalent to the distortion is applied individually to each loaded element, according to an axis of its local reference frame.

Conditions of use

- ❐ A distortion cannot be applied to an articulation element.

Examples

```
CAS DE CHARGE
'DISTORSION ux UNIFORMEMENT REPARTIE SUR DEUX ELEMENTS ENTIERS'
DISTORSION  ELEMENTS  REPARTIE  2
$  element n°    xr1   xr2     no_comp     q1     q2
$                                        (m/m)  (m/m)
      101       0.0   1.0        1       0.10   0.10
      102       0.0   1.0        1       0.10   0.10
```

Related commands

CAS DE CHARGE ; REDISTRIBUER

8.52 – [IDENTICAL] THERMAL LOADING

CHARGEMENT [IDENTIQUE] THERMIQUE *type_c* $[\left\{\begin{matrix}\text{POUTRE} & \text{no\_poutre} \\ \text{ELEMENTS} & \text{nb\_elements}\end{matrix}\right\}]$

$$\left\{\begin{matrix} xr_1 \quad xr_2 \quad V_1 \quad V_2 \\ \left\langle \text{no\_element} \quad xr_1 \quad xr_2 \quad V_1 \quad V_2 \right\rangle_{\text{nb\_elements}} \\ \left\langle \text{no\_element} \right\rangle_{\text{nb\_elements}} \quad xr_1 \quad xr_2 \quad V_1 \quad V_2 \end{matrix}\right\}$$

In this label, replace *type_c* by: $\left\{\begin{matrix}\text{DILATATION} & \\ \text{GRADIENT} & \left\{\begin{matrix}\text{LOCAL} & \left\{\begin{matrix}\text{Y}\\ \text{Z}\end{matrix}\right\} \\ \text{GLOBAL} & \theta_1 \quad \theta_2 \quad \theta_3\end{matrix}\right\}\end{matrix}\right\}$

Parameters

- θ_1, θ_2, θ_3: triple standardized rotation (see Figure 1.2) positioning a "'radiation" reference frame $OX_rY_rZ_r$, in which the O_rY_r axis indicates the direction and flow of the radiation, relative to the global reference frame (Figure 8.25).
- no_ptre: number of the beam to be loaded.
- nb_elements: number of elements to be loaded, positive.
- no_element: number of an element to be loaded, to be provided if the ELEMENTS option is used.

The thermal load described below applies to all the active elements of the beam no_poutre, to the element no_element, if the IDENTIQUE option is not used, or to the group of elements whose numbers precede it, if the IDENTIQUE option is used (this option does not agree with the ELEMENTS option).

By default, all non-articulation active elements are concerned for an expansion, and for a gradient, all the non-biarticulated and non-articulation elements.

- xr_1, xr_2: relative abscissas of the points of application of the start and end of the thermal load on an element, which is always assumed to be linearly distributed. These values are between 0.0 and 1.0 (including the boundaries) and expressed according to the x axis of its local reference frame, and xr_2 must be greater than xr_1.
- V_1, V_2: values of the thermal loading at the start and the end of the loaded zone. This is a difference in temperature if the DILATATION option is used, or a difference in temperature by unit of length (provided directly or evaluated, see the definition of the V_i parameter below) if the GRADIENT option is used.

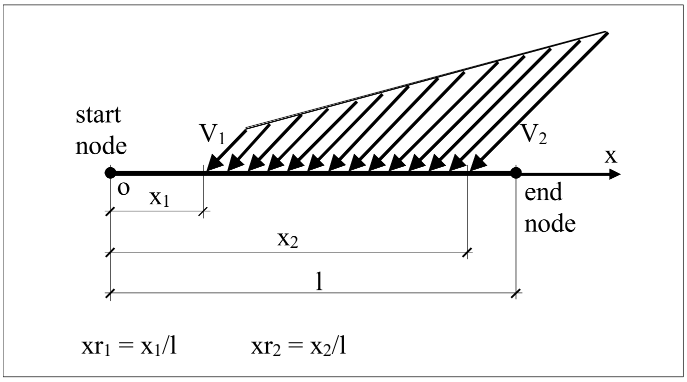


Figure 8. 32 - Linearly distributed thermal loading on an element

Functions

This command is used to apply a linearly distributed expansion or gradient thermal loading to all the active elements of the model, to all the active elements of a beam or to a group of elements, either individually or collectively.

The temperature gradient can be applied to each element according to the oy or oz axes of its local reference frame, or according to the OY axis of a unique arbitrary reference frame.

For an expansion applied to an element, V_i is the variation in temperature to which it is subjected, which is positive for an increase in length.

NOTE: For a temperature gradient applied to an element, V_i depends on the reference frame chosen to define it and, possibly, on the type of element. In particular, the GRADIENT LOCAL POUTRE option demands a difference in temperature.

The concrete Young modulus used for a thermal loading is the instant modulus calculated by PCP if the user entered the DATE command, and the reference modulus set by the user in the MATERIAU command otherwise.

LOCAL Y gradient (with POUTRE or ELEMENTS option)

$$V_i = \frac{dT}{dy} \qquad (8.3)$$

with:

- y: the ordinate in the local reference frame of the element,
- T: the corresponding temperature.

V_i is positive when the temperature increases with y.

LOCAL Z gradient (with POUTRE option)

$$V_i = Tz_e - Tz_i \qquad (8.4)$$

with:

- Tz_e: the temperature on the extrados,
- Tz_i: the temperature on the intrados.

xr_1 and xr_2 are usually 0.0 and 1.0, and the differences in values at the origin and the extremity of the element are calculated by the GE1 module, according to the o_iz_i axes of the generic reference frame of the corresponding sections (Figure 3.4), on the basis of the value attributed to the type_gth parameter of the GENERALITES command of the GE1 module (see Chapter 3).

V_i is positive when the temperature increases with z.

LOCAL Z gradient (with ELEMENTS option or without an option)

$$V_i = \frac{dT}{dz} \tag{8.5}$$

with:

- *z*: the value in the local reference frame of the element,
- *T*: the corresponding temperature.

V_i is positive when the temperature increases with z.

GLOBAL gradient (with POUTRE or ELEMENTS option or without an option)

The radiation effectively acts according to the OY_r axis of the $OX_rY_rZ_r$ reference frame.

The reference axis for the calculation of the thermal gradient is the intersecting straight line of the plane containing the ox axis of the element and an axis parallel to OY, with its straight section (oyz plane). Its direction is "opposite" to that of OY.

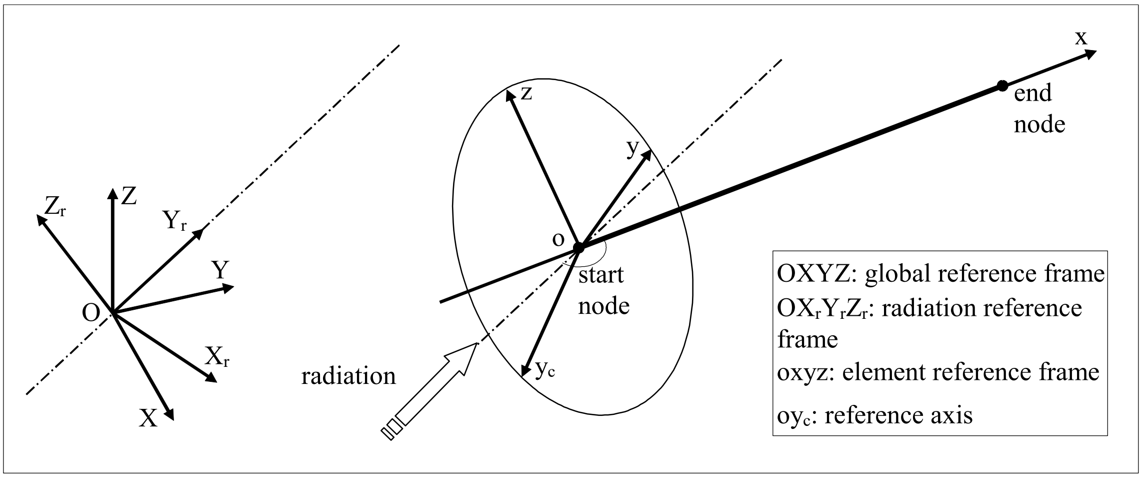


Figure 8.33 - Global thermal gradient applied to an element

$$V_i = \frac{dT}{dy} \tag{8.6}$$

with:

- *y*: ordinate measured according to oy_c;
- *T*: the temperature at the ordinate point *y*, calculated as if the radiation acted according to the inverted oy_c axis. Correction by the projection of OY_r on oy_c is automatic.

This type of loading eliminates the effects of the rotations of the local reference frames of the elements, irrespective of the direction of the radiation.

Conditions of use

- A gradient type loading cannot be applied to a biarticulated element.
- An articulation element cannot receive any thermal loading.

Methodological advice

- Using the global gradient type loading is simple when the radiation acts according to a vertical or horizontal axis. Precautions must be taken in any other case, when calculating the values of V_i.

Examples

```
CAS DE CHARGE
'AUGMENTATION DE TEMPERATURE UNIFORME DE 10 DEGRES'
$ applies by default to all the non-articulation active elements, because
$ no additional option is used, according to their longitudinal axes
CHARGEMENT  THERMIQUE  DILATATION
$ xr1     xr2     V1     V2
  0.0     1.0    10.0   10.0

CAS DE CHARGE
'AUGMENTATION DE TEMPERATURE LINEAIREMENT VARIABLE DE 0 A 10 DEGRES'
$ applies to a series of whole elements
CHARGEMENT  THERMIQUE  DILATATION  ELEMENTS  5
$ no_element     xr1     xr2     V1     V2
    1001         0.0     1.0     0.0    2.0
    1002         0.0     1.0     2.0    4.0
    1003         0.0     1.0     4.0    6.0
    1004         0.0     1.0     6.0    8.0
    1005         0.0     1.0     8.0   10.0

CAS DE CHARGE
'GRADIENT THERMIQUE UNIFORME DE 15 DEGRES SUR TABLIER 2'
$ applies to all active elements of a beam according to axes oizi of the
$ generic benchmarks of its sections; temperature of the upper structure
$ exceeds that of the lower structure by 15 degrees
CHARGEMENT  THERMIQUE  GRADIENT  LOCAL  Z  POUTRE  102
$ xr1     xr2     V1     V2
  0.0     1.0    15.0   15.0

CAS DE CHARGE
'GRADIENT THERMIQUE GLOBAL APPLIQUE SELECTIVEMENT'
$ applies to two series of elements along a vertical descending axis,
$ with two intensities: 5.0/2.0 and 5.0/4.0 (expressed in degrees per
meter)
CHARGEMENT IDENTIQUE THERMIQUE GRADIENT GLOBAL 0.0   0.0  -90.0 ELEMENTS
5
$                                            (teta1 teta2 teta3)
  1001 A 1005
$ xr1     xr2     V1     V2
  0.0     1.0    2.5    2.5
CHARGEMENT IDENTIQUE THERMIQUE GRADIENT GLOBAL 0.0   0.0  -90.0 ELEMENTS
5
  1006 A 1010
  0.0     1.0    1.25   1.25
```

Related commands

CAS DE CHARGE

8.53 – [AUTOMATICALLY] PUSH STRUCTURES

POUSSER [AUTOMATIQUEMENT] STRUCTURES *param_p*

In this label, replace *param_p* by:

$$\text{nb\_strpou}\quad \left\langle \text{nom\_strpou} \right\rangle_{\text{nb\_strpou}} \quad [\begin{Bmatrix} + \\ - \end{Bmatrix}]\text{nb\_elements}\quad [\ {}^{*}\begin{Bmatrix}\text{FOIS} & \text{nb\_fois} \\ \text{APPUIS} & \text{nb\_appuis}\end{Bmatrix}]$$

Parameters

- nb_strpou: the number of structures to be pushed simultaneously, positive.
- nom_strpou: the name of a structure to be pushed, defined on a DEFINIR STRUCTURE POUSSEE command.
- nb_elements: number of elements indicating the amplitude of the displacement, preceded by the *minus* sign if the push must be in the negative direction, or the *plus* sign, or no sign at all, if the push is made in the positive direction (see DEFINIR STRUCTURE POUSSEE command).
- nb_fois: a positive value indicating that the elementary push operation of an amplitude of nb_elements elements, must be repeated nb_fois times. To be provided if the AUTOMATIQUEMENT option is used.
- nb_appuis: number of temporary supports used with the AUTOMATIQUEMENT option. Positive, the default value is 2.

Functions

This command is used to displace a group of pushed structures in a single operation (NON AUTOMATIQUE mode), or to repeat this operation several times, while controlling the installation of the supports and links, the editions and the calculation of the extreme values (AUTOMATIQUE mode).

NON AUTOMATIQUE mode (without the AUTOMATIQUEMENT option)

All the active elements of the selected pushed structures are displaced simultaneously, along their respective paths.

Please note that this displacement is not enough to simulate a push operation, because the conditions at the boundaries remain unchanged.

First, the active links between the pushed structures and the rest of the model must be removed, and then subsequently restored (see Annex D, simulation of a push operation).

The deformed structure and the supports linked to the pushed element follow the displacement. In this way:

- the row i element of each pushed structure occupies the incidence of its row i element–nb_elements or i+nb_elements (depending on the direction of the push operation),
- the support that may be placed at its origin or its end node will be moved in the same manner, while keeping its relative position,
- the displacements of its incident nodes will be applied to the incident nodes of its host element.

The positions of the active nodes of the structures pushed by TRANSLATION or ROTATION (see the DEFINIR STRUCTURE POUSSEE command) may be altered. This is the case of constant height caissons with cores or slabs with variable thicknesses (see Annex D, simulation of a push operation).

For structures pushed by ROTATION or EMBOÎTEMENT, the displacement modifies the rigidity of the element (calculated in the global reference frame).

A rotation also applies to the displaced supports. When a rotation (for example, of a non-vertical axis) does not keep the deformations of elements under gravitational loads, the PH3 module removes, before the push operation, the dead weight of the elements declared to be heavy, and restores it after the push operation.

AUTOMATIQUE mode (without the AUTOMATIQUEMENT option)

Several push phases can be simulated with a single command.

After finding the active links between the pushed structures and the rest of the model, the following operations are performed nb_fois times:

- placement of nb_appuis temporary supports on the pushed structures,
- possible inhibition of the editions and the calculation of the extreme values,
- removal of all the links,
- displacement of the pushed structures with nb_elements elements (this operation is equivalent to a push operation in NON AUTOMATIQUE mode),
- placement of the links connected to the displaced structures, with the total reestablishment of continuity,
- possible reactivation of the editions and the calculation of the extreme values,
- removal of the temporary supports,
- placement of the links not connected to the pushed structures.

After these operations, the pushed structures have undergone a displacement of nb_fois*nb_elements elements, and all the articulations that were active before the push operation remain active.

When an element reaches a support without resting on it, and then resting on it, the corresponding link must be activated after the push operation, and the next push operation must be performed in AUTOMATIQUE mode, with zero amplitude.

Conditions of use

- ☐ This command can only be used in CONCORDANCE SUSPENDUE and CUMUL modes.
- ☐ Structures that are pushed simultaneously must be defined as being of the same type (DROIT, TRANSLATION, ROTATION or EMBOÎTEMENT), and with compatible parameters.
- ☐ No isolated supports must be on the push paths during the push operation.
- ☐ The pushed structures cannot cross one another at a given node.
- ☐ The active elements of the pushed structures must never be linked to active elements that do not belong to the pushed part of the model.
- ☐ All the active elements of the pushed structures must have been displaced before the push operation by the PLACER ELEMENTS commands with the DEPLACES option.

- nb-elements and possibly nb_fois must be compatible with the current positions of the active elements on their push paths, which they must not leave, neither at the start, not at the end.
- Using the AUTOMATIQUE mode demands the presence in the model of at least one support-type that is perfectly blocked according to the six degrees of freedom (see the PARFAIT option of the APPUI command of the PH1 module, Chapter 6).
- The AUTOMATIQUE mode cannot be used if continuity must only be partially reestablished (when placing the links connected to the displaced structures).

Methodological advice

- See Annex D, simulation of a push operation.

Examples

The structure below comprises two beams with rectilinear mean fibers, linked by transverse elements, and three pushed structures.

In the construction phase in question, the links, modeled by articulation elements and supports, are in place in the prefabrication zone (which includes an abutment) and on the first pile, and a group of elements corresponding to two "sections" of the structure have just been displaced towards the prefabrication zone and activated.

This assembly, made up of two elements, is to be pushed in NON AUTOMATIQUE mode in the positive direction.

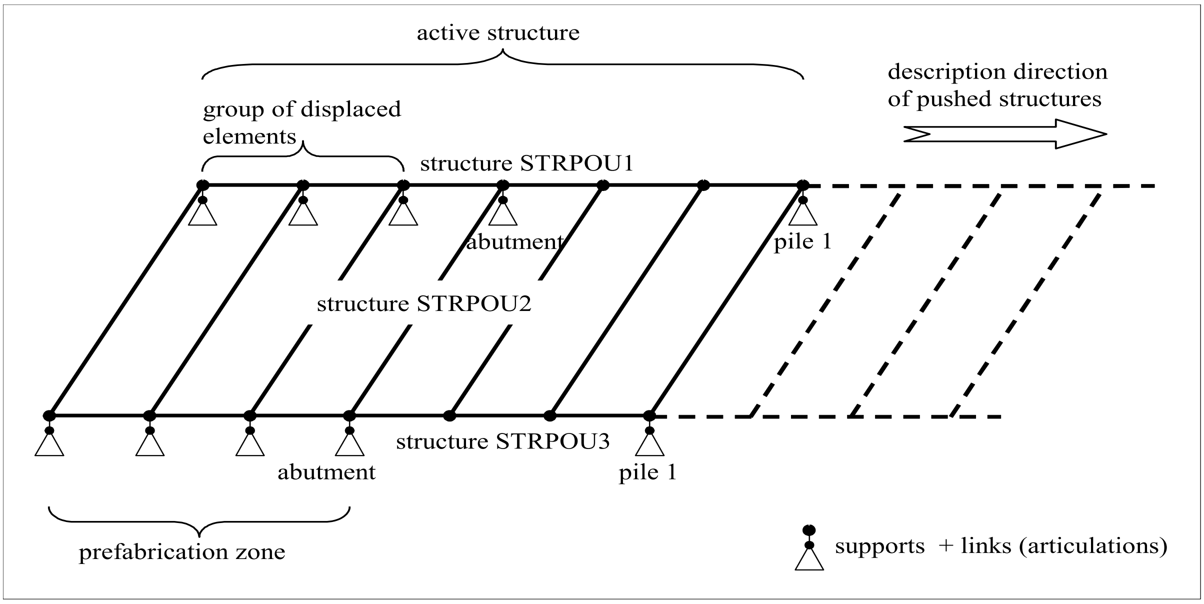
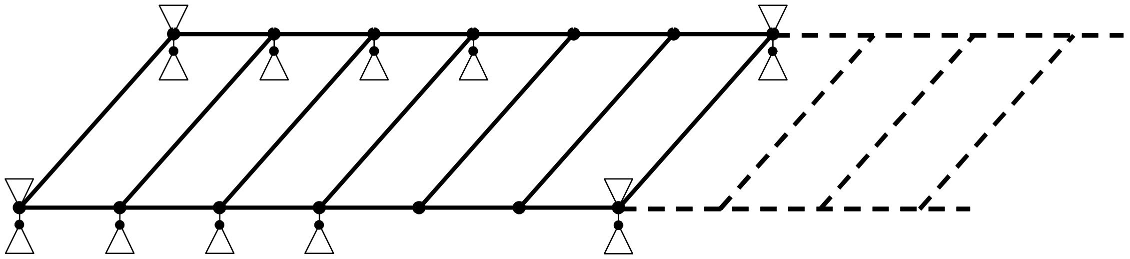
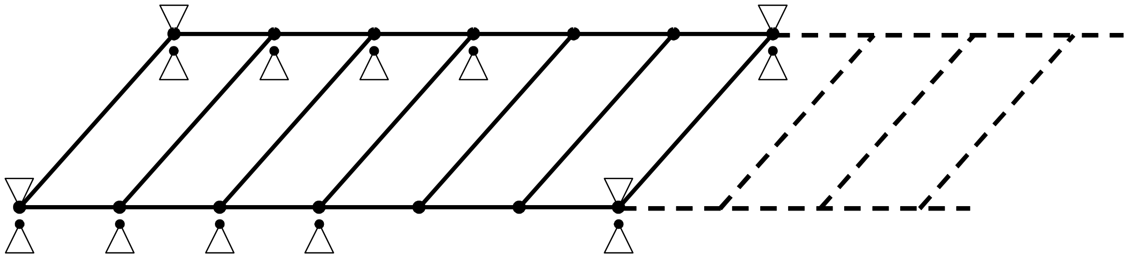


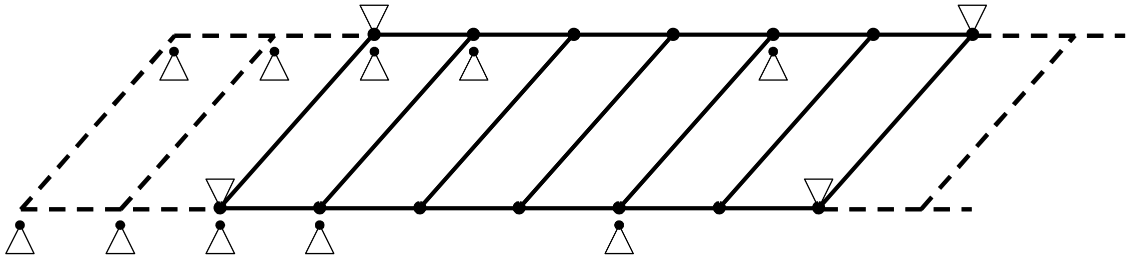
Figure 8.34 - Structure to be pushed: initial state



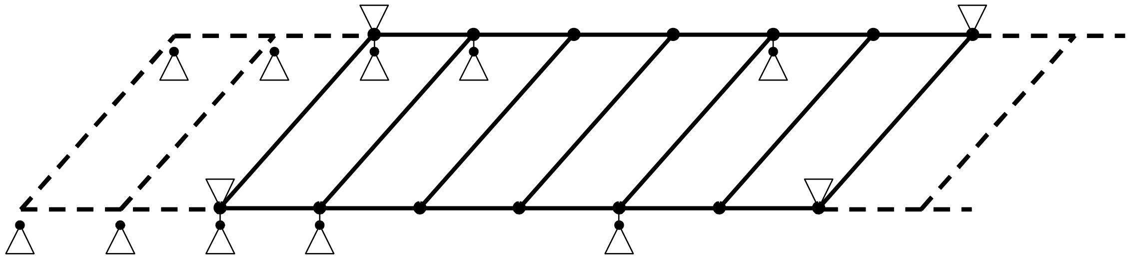
step 1: placement of temporary supports on structures to be pushed



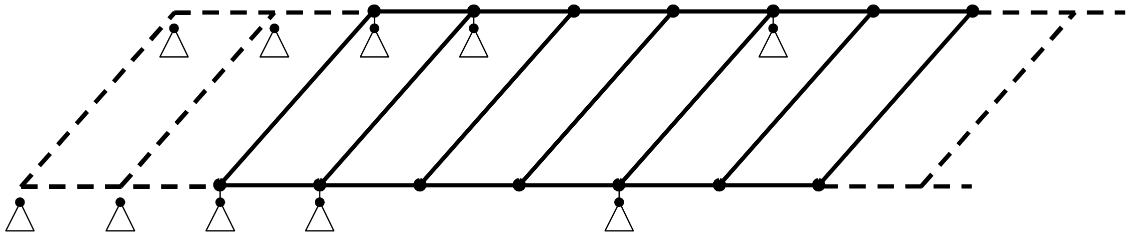
step 2: removal of links



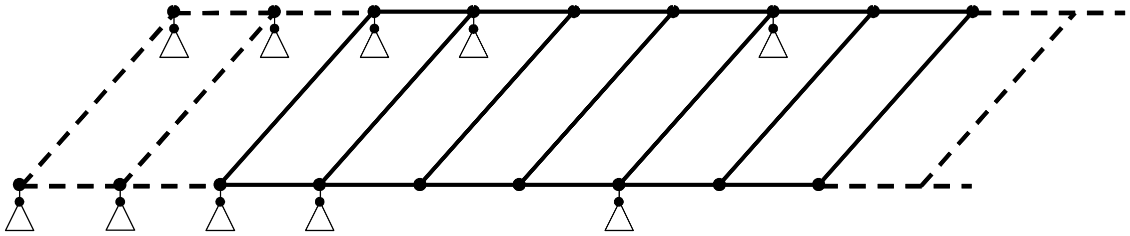
step 3: two elements pushed in the positive direction



step 4 : placement of links connected to the pushed structures, with reestablishment of continuity,



step 5: removal of temporary supports



step 6: placement of links not connected to the pushed structures

Figure 8.35 - Pushing two elements

```
PHASES
..............
CALCULER  EXTREMAS
EDITER  ...
..............
DEFINIR  STRUCTURE  POUSSEE  STRPOU1  ...
..............
DEFINIR  STRUCTURE  POUSSEE  STRPOU2  ...
..............
DEFINIR  STRUCTURE  POUSSEE  STRPOU3  ...
..............
SUSPENDRE  CONCORDANCE
$ current push phase
PLACER  ELEMENTS  DEPLACES  6
..............
ACTIVER  ELEMENTS  6
..............
$ step 1
$ -------
PLACER  APPUIS  4
..............
SUSPENDRE  EDITIONS
SUSPENDRE  ANALYSE
$ step 2
$ -------
SUPPRIMER  ELEMENTS  10
..............
$ step 3
$ -------
POUSSER  STRUCTURES  3  STRPOU1  STRPOU2  STRPOU3  2
$ step 4
$ -------
PLACER  ARTICULATIONS  CONTINUITE  6
..............
CONTINUER  EDITIONS
CONTINUER  ANALYSE
$ step 5
$ -------
SUPPRIMER  APPUIS  4
..............
$ step 6
$ -------
PLACER  ARTICULATIONS  4
..............
..............
```

Related commands

VERIFIER ; DEFINIR STRUCTURE POUSSEE ; CALCULER EXTREMAS

SUSPENDRE / CONTINUER CONCORDANCE ; EDITER / NONEDITER

DATE ; PLACER APPUIS ; SUPPRIMER APPUIS ; PLACER ELEMENTS

PLACER ARTICULATIONS ; ACTIVER ELEMENTS ; SUPPRIMER ELEMENTS

ETAT

8.54 - REDISTRIBUTE

REDISTRIBUER v_2

Parameters

- v_2: coefficient of redistribution applicable to the loaded structure, in its current static condition. Must be positive and less than 1.0.

Functions

This command is used to simulate a redistribution of the solicitations by changing the static conditions or by displacements imposed on the supports, in CALCUL FORFAITAIRE mode, and under certain conditions (see Annex D). The following operations are performed:

- weighting of the current state by $1.0 - v_2$,
- weighting of the loads applied to the nodes, accumulated since the start of the construction, by v_2,
- application of this loading to the active structure and calculation of its new state of deformation and solicitation.

Please note that the effect produced only contains the part relative to this loading, corresponding to the solicited structure in its current static condition, whereas as the state obtained contains the two components of the redistribution.

The redistribution operation is linear relative to v_2. Therefore, for a given coefficient *k* and an operator *S(o)*, representing the state obtained after the operation *o*, we have:

$$S(redistribu\ tion(k \cdot v_2)) = S(k \cdot redistribu\ tion(v_2)) \tag{8.7}$$

Conditions of use

- ❐ This command cannot be introduced in CALCUL NON LINEAIRE mode nor in CALCUL RHEOLOGIQUE FIN mode.
- ❐ It is not possible to simulate a redistribution after tensioning or distorting elements.

Methodological advice

- ❐ See Annex D, flat-rate calculation.

Examples

```
AFFECTER  CONTRAINTES  UNIQUES  680.0
..............
TITRE 'OUVRAGE APRES REDISTRIBUTION'
REDISTRIBUER  0.5
```

Related commands

VERIFIER ; EDITER / NONEDITER ; AFFECTER CONTRAINTES

DATE ; TENDRE ELEMENTS ; DISTORSION ELEMENTS ; ETAT

8.55 - SAVE

SAUVER nom_structure

Parameters

- nom_structure: the name under which the current structure will be saved and called.

Functions

This command records the current structure in the database, including:

- the complete description of its static conditions (placed supports and articulations, active elements) and its rigidity matrix,
- the state of progress of certain operations (cables tensioned, cables injected, form travelers in place),
- certain additional data that has been introduced (title, elastoplastic materials, chains, pushed structures, footing of the form traveler),
- the selected calculation and operating options (non linear calculation, losses of prestress, optimization, stress calculation, extreme values calculation, recording, concordance, transfer of deformations, accumulation mode, rheological calculations), excluding the edition options,
- its probable state of deformation and solicitation.

All these options can be used:

- to interrupt a construction process and resume it in a later session (using the SUITE option of the PHASES command),
- to keep an "intermediate" structure that can be used again later, and to apply certain loadings to it,
- to call the saved structure and the associated tangential rigidity in order to calculate the surfaces of influence and the envelope effects under the operating loads, using the ENV module,
- to call the saved structure in order to calculate the dynamic effects due to the wind or earthquakes, using the DYN module,
- the retrieval by the ETU module of its probable state, its weighting, its edition or its integration in combinations or envelopes. This is usually recorded under the number 0 and can be called using the name: « ETAT 0 STRUCTURE nom_structure » ;
- the extraction by the ETU module of certain beam sections, with their possible prestress, in order to prepare their verification in the boundary states using the CDS software (according to the current tensions in the cables).

The reference title of a saved structure is the title that appears on the last TITRE command that precedes the corresponding SAUVER command.

Conditions of use

- ❒ This optional command can be introduced several times, but only after starting the construction process.
- ❒ A structure saved in CONCORDANCE SUSPENDUE mode cannot be used in calculations of envelopes by the ENV module.
- ❒ It is not authorized to save a structure when the analysis of the extreme values is suspended.
- ❒ A saved structure is replaced in the database when a request is made to record it again, and any surfaces of influence that are attached to it are deleted.
- ❒ The total number of saved structures is limited to 100.
- ❒ The title must be updated before saving a structure in order to indicate its nature and content.

Examples

```
PHASES
EDITER  ...
DATE  CONSTRUCTION  100
TITRE 'OUVRAGE EN SERVICE'
SAUVER  STRUSERV
DATE  CONSTRUCTION  200
$ simulation of aging, without changing the static scheme
..............
DATE  CONSTRUCTION  10000
ETAT  1
FIN
```

At the end of this session, the database contains:

- the saved structure STRUSERV, called 'OUVRAGE EN SERVICE';
- its probable state on the dates 100 and 10000, which can be called by the ETU module under the following names:
  - ETAT 0 STRUCTURE STRUSERV ;
  - ETAT 1 STRUCTURE STRUSERV.

Retrieval of the saved structure in order to apply certain isolated load cases:

```
PHASES  SUITE  STRUSERV
$ the unsaved edition options must be recalled
EDITER  ...
..............
CAS DE CHARGE
..............
CAS DE CHARGE
..............
FIN
```

Related commands

PHASES ; VERIFIER ; TITRE ; SUSPENDRE / CONTINUER CONCORDANCE

SUSPENDRE / CONTINUER ANALYSE ; EDITER / NONEDITER ; DATE

CAS DE CHARGE ; ETAT

8.56 - STATE

$$\text{ETAT}\quad [\left\{\begin{array}{l}\text{EXTREMAL}\\ \text{PONDERE}[\;{}^{*}\left\{\begin{array}{l}\text{KGFAVORABLE}\;\; v_1\\ \text{KGDEFAVORABLE}\;\; v_2\\ \text{KPOFAVORABLE}\;\; v_3\\ \text{KPODEFAVORABLE}\;\; v_4\\ \text{KPEFAVORABLE}\;\; v_5\\ \text{KPEDEFAVORABLE}\;\; v_6\\ \text{KPTFAVORABLE}\;\; v_7\\ \text{KPTDEFAVORABLE}\;\; v_8\end{array}\right\}]\\ \text{PRECONTRAINT}\end{array}\right\}]\quad \text{no\_etat}$$

Parameters

- ν_1, ν_2, ν_3, ν_4, ν_5, ν_6, ν_7, ν_8: weighting coefficients of the favorable and unfavorable effects of the permanent loads (excluding prestress), of the initial tensions of the cables and of the losses of prestress for the calculation of a weighted state. Any values that are not provided are fixed at 1.0 by default for ν_1, ν_2, ν_3, ν_4, ν_5 and ν_6, and at 0.0 by default for ν_7 et ν_8. The coefficients ν_3, ν_4, ν_5 and ν_6 must not appear when ν_7 and ν_8 are fixed, in this case, ν_3, ν_4, ν_5 and ν_6 are null; and vice versa, ν_7 and ν_8 must not appear when ν_3, ν_4, ν_5 and ν_6 are fixed.
- no_etat: identification number of the state to be recorded in the database. Positive and less than 90_000.

Functions

This command is used to record the following information in the database:

- the state of deformation and solicitation of the active structure (PROBABLE state is the default option),
- its EXTREMAL state of solicitation (EXTREMAL option),
- a PONDERE envelope state of solicitation (PONDERE option),
- or its PRECONTRAINT state, which is a state of solicitation reduced to the isostatic effects of the prestress (PRECONTRAINT option).

The PONDERE and the PRECONTRAINT states are edited.

The recorded state can be extracted by the ETU module in order to be weighted, edited or integrated in a combination or an envelope. It can also be viewed by the RES module.

It is attached to the last restored structure (SUITE option of the PHASES command) or is saved (SAUVER command) later on in the same session, if one exists. Otherwise it is identified by its number only.

The reference title of a recorded state is the title that appears on the last TITRE command that precedes the corresponding ETAT command.

PROBABLE state

The probable state is obtained when no options of the type (EXTREMAL, PONDERE or PRECONTRAINT) are selected. This state covers the following effects, which are accumulated from the start of the construction:

- the displacements of the nodes of the active structure,
- the reactions of supports in their local reference frames,
- the forces, in local reference frames, of all types of active elements,
- the forces, in section reference frames, of the active beam elements,
- the normal and tangential stresses in the active beam elements (only in CALCUL CONTRAINTES mode).

The list of active elements and the list of placed supports are recorded together.

EXTREMAL state

This is an envelope state that contains the extreme values (minimum and maximum values, usually distinct, that occurred during the construction) of the effects (and components) selected in the various preceding CALCULER EXTREMAS commands.

PONDERE state

This is an envelope state that contains the minimum and maximum values calculated for the effects (and components) selected in the various preceding CALCULER EXTREMAS commands. This command does not apply to the calculation of displacement.

It is used to address various calculations in values that are *characteristic* of the prestress forces, as recommended by certain regulations, and can be established in CALCUL FORFAITAIRE mode or in CALCUL RHEOLOGIQUE FIN mode, under certain conditions (see conditions of use and Annex D, characteristic prestress).

It is obtained by combining the following effects (accumulated from the start of the construction).

- the effect of the implicit and explicit loadings (introduced in CUMUL mode), other than that of the prestress, weighted by the coefficient ν_1 if it is favorable (KGFAVORABLE option), or by the coefficient ν_2 if it is unfavorable (KGDEFAVORABLE option).
- the effect of the initial tensions of the cables, weighted by the coefficient ν_3 if it is favorable (KPFAVORABLE option), or by the coefficient ν_4 if it is unfavorable (KPDEFAVORABLE option).
- the effect of the prestress losses, weighted by the coefficient ν_5 if it is favorable (KPEFAVORABLE option), or by the coefficient ν_6 if it is unfavorable (KPEDEFAVORABLE option).
- The effect of the global prestress, that is to say the initial tensions of the cables and the prestress losses, weighted by the coefficient ν_7 if it is favorable (KPTFAVORABLE option), or by the coefficient ν_8 if it is unfavorable (KPTDEFAVORABLE option), according to the Eurocode 2-1-1, paragraph 5.10.9.

PRECONTRAINT state

This is the state of forces (in element and section reference frames) and stresses (normal and tangential) in the active beam elements, corresponding to the *isostatic* effects of the prestress, accumulated from the start of the construction.

Conditions of use

- This optional command can be introduced several times, but only after starting the construction process.
- The identification number of an EFFET already recorded by the PH3, ENV, DYN or ETU modules (according to the terminology used by the ETU module, see Table 12.1) cannot be used to designate a recorded STATE.
- A recorded state is replaced in the database (irrespective of its type) when it is recorded again with the same number.
- The number of states that can be recorded is unlimited, apart from their identification numbers.
- The current title must be updated before recording a state in order to indicate its nature and content.
- Attaching several states to a saved structure is purely formal and can correspond to different static situations.
- An EXTREMAL or PONDERE state can only be requested if at least one CALCULER EXTREMAS command has already been introduced.
- A PONDERE or PRECONTRAINT state can only be requested if if the model has at least one cable.
- A PONDERE state can only be requested if the previous construction process has at least:
    - one placement of a support with jacking or one removal of a support,
    - one replacement of a support, with or without jacking,
    - one placement of an articulation with reestablishment of continuity,
    - one replacement of an articulation,
    - one removal or one tensioning of an element,
    - one push operation in CALCUL RHEOLOGIQUE FIN mode.
- For push operations in CALCUL FORFAITAIRE mode, the ETAT PONDERE command must be preceded by a REDISTRIBUER command.

Methodological advice

- If we are only interested in the results (to be processed by the ETU module or viewed by the RES module), the ETAT command must be preferred to the SAUVER command, which also records a large quantity of construction data in a given phase.
- According to the combinations described in the Eurocodes, the use of the PONDERE state with the same value of ν_3 and ν_5 is not equivalent to the use of this state with the same value for ν_7. Similarly, the use of the PONDERE state with the same value of ν_4 and ν_6 is not equivalent to the use of this state with the same value for ν_8.

Examples

Update of a PROBABLE state.

```
PHASES
$ session 1 first variant of the construction process
..............
$ updated current title before saving the current state
$ (for identification)
TITRE  'ETAT PROBABLE DE L''OUVRAGE A SA MISE EN SERVICE'
$ the ETAT command without a type option designates a PROBABLE state
$ since no structure was previously restored or saved,
$ it is only identified by its number
ETAT  1
FIN

PHASES
$ session 2 second variant of the construction process
..............
TITRE  'ETAT PROBABLE DE L''OUVRAGE A SA MISE EN SERVICE'
$ the new state PROBABLE replaces the old one in the database
ETAT  1
FIN
```

Recording a series of PROBABLE states, some of which are attached to a saved structure.

```
PHASES
$ session 1 simplified construction process of the structure
$ supposed to be cast right away on an arch
..............
TITRE  'ETAT PROBABLE DE L''OUVRAGE COULE SUR CINTRE'
ETAT  1
FIN

PHASES
$ session 2 detailed construction process
..............
TITRE  'ETAT PROBABLE DE L''OUVRAGE A SA MISE EN SERVICE'
$ this command saves the construction data and the
$ corresponding PROBABLE state number 0
SAUVER  STRUSERV
..............
TITRE 'ETAT PROBABLE DE L''OUVRAGE A 1000 JOURS'
DATE  VIEILLISSEMENT  ETAT  1000
ETAT  1
TITRE 'ETAT PROBABLE DE L''OUVRAGE AU TEMPS INFINI'
DATE  VIEILLISSEMENT  INFINI  ETAT
ETAT  2
FIN
```

At the end of these two sessions, the database contains four PROBABLES states that can be called by the ETU module under the following names:

- ETAT 1 ;
- ETAT 0 STRUCTURE STRUSERV ;
- ETAT 1 STRUCTURE STRUSERV ;
- ETAT 2 STRUCTURE STRUSERV.

Recording different types of states: PROBABLES, PONDERES, PRECONTRAINTS and EXTREMAL, attached to a saved structure.

```
PHASES
$ calculation of the stresses in the beam elements
CALCULER  CONTRAINTES
$ at least one command of this typs is necessary to calculate the
$ PONDERES and EXTREMAUX  states
CALCULER  EXTREMAS  CONTRAINTES  NORMALES  COMPOSANTE  1
..............
TITRE  'ETAT PROBABLE DE L''OUVRAGE A SA MISE EN SERVICE'
SAUVER  STRUSERV
TITRE  'ETAT CARACTERISTIQUE DE L''OUVRAGE A SA MISE EN SERVICE'
$ this state contains the minimum and maximum values of the normal
$ stresses (SIGMA) in the sections of the active beam elements
$ the effect of the accumulated permanent loads (excluding weighting) is
$ not weighted (nu1 = 1.0 and nu2 = 1.0, by default)
$ the favorable effect of the initial cable tensions is weighted
$ by 0.98, and their unfavorable effect by 1.02
$ the favorable effect of the losses of pre-stress is weighted
$ by 0.80, and their unfavorable effect by 1.20
ETAT  PONDERE  KPOFAV 0.98  KPODEF 1.02  KPEFAV 0.80  KPEDEF 1.20  1
TITRE 'ETAT PRECONTRAINT DE L''OUVRAGE A SA MISE EN SERVICE'
$ this state contains the forces and stresses in the active beam
$ elements, due to the accumulation of isostatic pre-stress loads
ETAT  PRECONTRAINT  2
..............
TITRE  'ETAT PROBABLE DE L''OUVRAGE AU TEMPS INFINI'
DATE  VIEILLISSEMENT  INFINI  ETAT
ETAT  10
TITRE  'ETAT CARACTERISTIQUE DE L''OUVRAGE AU TEMPS INFINI'
ETAT  PONDERE  KPOFAV 0.98  KPODEF 1.02  KPEFAV 0.80  KPEDEF 1.20  11
TITRE  'ETAT PRECONTRAINT ISOSTATIQUE DE L''OUVRAGE AU TEMPS INFINI'
ETAT  PRECONTRAINT  12
TITRE  'ETAT EXTREMAL DE L''OUVRAGE AU TEMPS INFINI'
$ this state contains the envelope values of the normal stresses
$ in the sections of the active beam elements
ETAT  EXTREMAL  20
FIN
```

At the end of this session, the database contains seven states of various types, which can be called by the ETU module under the following names:

- ETAT 0 STRUCTURE STRUSERV (PROBABLE) ;
- ETAT 1 STRUCTURE STRUSERV (PONDERE) ;
- ETAT 2 STRUCTURE STRUSERV (PRECONTRAINT) ;
- ETAT 10 STRUCTURE STRUSERV (PROBABLE) ;
- ETAT 11 STRUCTURE STRUSERV (PONDERE) ;
- ETAT 12 STRUCTURE STRUSERV (PRECONTRAINT) ;
- ETAT 20 STRUCTURE STRUSERV (EXTREMAL) .

Related commands

VERIFIER ; TITRE ; CALCULER EXTREMAS ; AFFECTER CONTRAINTES ; DATE; PLACER APPUIS ; REMPLACER APPUIS ; SUPPRIMER APPUISPLACER ARTICULATIONS ; REMPLACER

ARTICULATIONS ; TENDRE ELEMENTS ; SUPPRIMER ELEMENTS ; CAS DE CHARGE ; POUSSER [AUTOMATIQUEMENT] STRUCTURES ; REDISTRIBUER ; SAUVER

8.57 – PRINT EXTREME VALUES

IMPRIMER EXTREMAS

Functions

This command edits the current extreme state for the effects (and components) selected in all the CALCULER EXTREMAS commands that precede it.

Conditions of use

- ❐ This optional command can be introduced several times, but only after starting the construction process.
- ❐ It must be preceded by at least one CALCULER EXTREMAS command.

Examples

```
PHASES
CALCULER  EXTREMAS  CONTRAINTES  NORMALES  COMPOSANTE  1
..............
$ edition of the minimal and maximum values of the normal stresses
$ occurring in the active beam elements, since the start of the
$ construction, under the effect of all the implicit or explicit
$ loads applied in CUMUL mode
IMPRIMER  EXTREMAS
..............
```

Related commands

VERIFIER ; TITRE ; CALCULER EXTREMAS

8.58 – PRINT TENSIONS

IMPRIMER TENSIONS [nb_cables]

$$\langle \text{nom\_cable} \rangle_{\text{nb\_cables}}$$

Parameters

- nb_cables: the number of tensioned cables, whose tension we want to print, positive.
- nom_cable: name of a selected cable.

The names of the cables are only provided if nb_cables is also provided. Otherwise, all the tensioned cables will be printed out, by default.

Functions

This command is used to print out the current tensions in a group of cables, at all the beam sections that they cross.

Conditions of use

- ❐ This optional command can be introduced several times, in various steps of the construction.
- ❐ It can only be used if the model has at least one tensioned cable.
- ❐ It can be used in CALCUL NON LINEAIRE, CALCUL FORFAITAIRE or CALCUL RHEOLOGIQUE FIN mode.

Examples

```
TENDRE  CABLES  2
FL101G  FL101D
..............
TENDRE  CABLES  2
FL102G  FL102D
..............
TENDRE  CABLES  2
FL103G  FL103D
..............
$ edition of the tensions in all the taut cables
IMPRIMER  TENSIONS
$ edition of the tensions in two selected cables
IMPRIMER  TENSIONS  2
FL101G  FL102G
..............
```

Related commands

VERIFIER ; TITRE ; TENDRE CABLES ; DETENDRE CABLES

8.59 – CRITICAL ANALYSIS

ANALYSE CRITIQUE MODES nb_pre_modes [${}^{*}\left\{\begin{array}{ll} \text{TOLERANCE} & \text{toler} \\ \text{ITERATIONS} & \text{nb\_iter} \\ \text{ENREGISTRER} & \text{no\_pre\_mode} \end{array}\right\}$]

Parameters

- nb_pre_modes: the number of requested buckling modes, positive.
- toler, nb_iter: the tolerance threshold and maximum number of iterations to be used in the evaluation of and the search for buckling modes (1.0E-6 and 100 by default).
- no_pre_mode: record number of the first mode. The modes are recorded in a database, if necessary, with the numbers no_pre_mode à no_pre_mode + nb_pre_modes – 1, which must remain positive and less than 90_000.

Functions

The critical analysis consists in studying the linear buckling of a structure loaded in one or several steps.

This command starts the calculation of an imposed number of buckling modes of the structure being studied. These "first modes" are numbered consecutively from one (order numbers).

This instability calculation consists in establishing the critical load factor (relative to all the applied loads) associated with each of the required modes, which are of the unit standard, as per the infinite standard, and which can be edited as EFFETS by the PH3 module.

With the ENREGISTRER option, they are recorded in the database (such that the maximum displacement equals 1) with a series of consecutive numbers (usually different from their order numbers), for which the starting value is provided. These are MODES type EFFETS that can be invoked by the ETU or RES modules. By default, no modes are recorded.

At the end of the critical analysis, the deformation corresponding to a buckling mode can be introduced as an initial geometric fault, using the MODE option of the CALCUL NONLINEAIRE command.

Conditions of use

- This optional command can be introduced several times, in various steps of the construction.
- The search for buckling modes must be made in CALCUL NON LINEAIRE mode.
- The substructure being studied must be loaded beforehand.
- The identification number of an EFFET already recorded by the PH3, ENV, DYN or ETU modules (according to the terminology used by the ETU module, see Table 12.1) cannot be used to designate a recorded MODE of buckling.

Methodological advice

- A critical analysis of linear buckling is only an approximative study of the stability of form.

- An incremental analysis can adopt a more precise approach to the phenomenon by applying a load that exceeds the critical threshold and by looking for the ultimate load.

Examples

```
PHASES
CALCUL  NONLINEAIRE
..............
$ the tolerance threshold set by default (1.0E-6) is suitable
$ the first three modes of buckling will be saved
$ in the database with the numbers 150, 151 and 152
ANALYSE  CRITIQUE  MODES  3  ITERATIONS  50  ENREGISTRER  150
```

Related commands

VERIFIER ; CALCUL NONLINEAIRE

8.60 - END

FIN

Functions

This command causes the end of a session and stops the execution of the PH3 module; all potential following commands are ignored.

Examples

```
PHASES
..............
..............
FIN
```

Related commands

PHASES

Chapter 9

Operating loads

Introduction

Appendix A provides the full wording of the document referenced in this chapter under the condensed name: "Issue 61", "Circular 83" and "Eurocode 1".

In this chapter, the term "beam" means a spatial beam.

Functions of the ENV module

This module is used to calculate the envelope values of the support reactions, forces and stress generated by the fixed or mobile loads, relative to certain given surfaces.

These operating loads may be applied to any structure activation pattern saved in the PH3 module during the construction. If the structure is saved in a non-linear calculation, the tangential rigidity matrix is used.

The ENV module can perform the following main operations:

- definition and recording of any number of areas of study, characterized by their type (displacements, support reactions, forces or stress to be calculated), and their location (selection of nodes or ends of elements);
- definition and recording of any number of loading surfaces, known as SUPPORTS, linked to sets of nodes or elements that receive loads (load-bearing structures), that may have CIRCULATION zones;
- application to these supports of fixed or mobile loading systems, called ACTIONS, to be simply selected, if they are predefined (the gravitational part of the main loads in Issue 61, Circular 83 or Eurocode 1), or to be defined using specific commands;
- calculation and recording of the ENVELOPPE effects produced, for various domains of study, by certain actions applied to certain supports.

The envelope curves can be edited or combined using the ETU module, either with one another, or with compatible results produced by the PH3 or DYN modules, or supplied by other means. The surfaces of influence, which are recorded as intermediate results, and the envelope curves can be viewed using the RES module.

The envelopes are calculated by the TRAFIC sub-module, which provides a language to describe the very general operating loads. All the operating loads that are not predefined (railway loads, etc.) can be simulated using this language, and must be stored in the file with the name: *trafic.don*.

Terminology

Area of study

An area of study is defined by its type and its location on the structure.

Type	Location on the structure
Displacements (*)	Selection of nodes
Support reactions	Selections of nodes having received a support
Forces in element reference frames	Selection of beam or non-beam element ends
Forces in section reference frames	Selection of beam element ends
Normal stress (**)	Selection of beam element ends
Tangential and normal stress	Selection of beam element ends

Table 9.1 - Areas of study

(*) A "displacements" type area of study cannot give rise to calculations of envelopes under operational loads, but can be used by the ETU or RES modules.

(**) Normal stress is calculated in homogeneous sections.

Support

The load-bearing structure of a support is an ordered list of nodes, in which the applied loads are taken into consideration.

The loading surface includes a reference AXE, of which each point of definition A_i has a local reference frame $A_i x_i y_i z_i$ (called the "axis reference frame") and FILES, that are only defined by their points of intersection with all the $A_i y_i z_i$ planes, in these reference frames (load application points), a line corresponding to the axis can be defined.

The longitudinal discretization of the loading surface must be linked to the distribution of the nodes of the load-bearing structure.

By way of example, the load-bearing structure may be assimilated to the mean fiber of a beam, since its reference axis is the same as its reference fiber (common case).

The local y coordinates of the points of a line are assumed to be constant, while the local z coordinates may vary.

Each reference frame linked to a point on the reference axis is a loading reference frame of the support, in the axes for which the following components can be applied, while the direction of the axes is respected:

- 1: force Fx applied according to $A_i x_i$,
- 2: force Fy applied according to $A_i y_i$,
- 4: force Fz applied according to $A_i z_i$, but in the opposite direction.

The component number 3 is reserved for the application of gravitational loads in the OZ axis of the global reference frame, but in the opposite direction (Figure 9.1).

The points defining the lines form a "mesh" that constitutes the model of the loading surface. All loads applied to a point of this mesh are transmitted in full to the node(s) of the corresponding load-bearing structure.

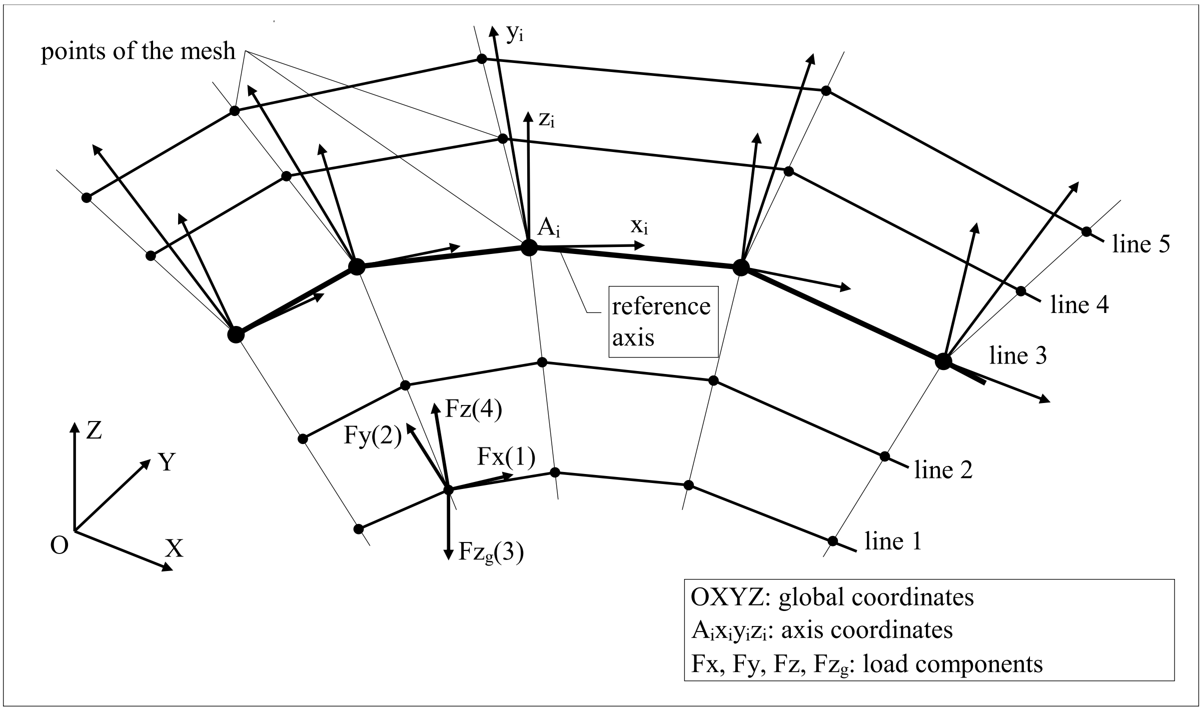


Figure 9.1 - Support with a line corresponding to the axis. Real model

At a given point of study, the calculation of the value of the influence of a load applied to each point of the mesh allows the corresponding surfaces of influence to be generated. This calculation takes account of the exact positions of the points of the mesh relative to the nodes of the load-bearing structure.

The search for envelopes requires the conversion of the real model, which is usually neither flat nor orthogonal, into a model that possesses these two properties. To this end, the ENV module develops the axis longitudinally, and corrects the transverse profile of the loading surface by projecting its lines according to the local z direction (Figures 9.2 and 9.3).

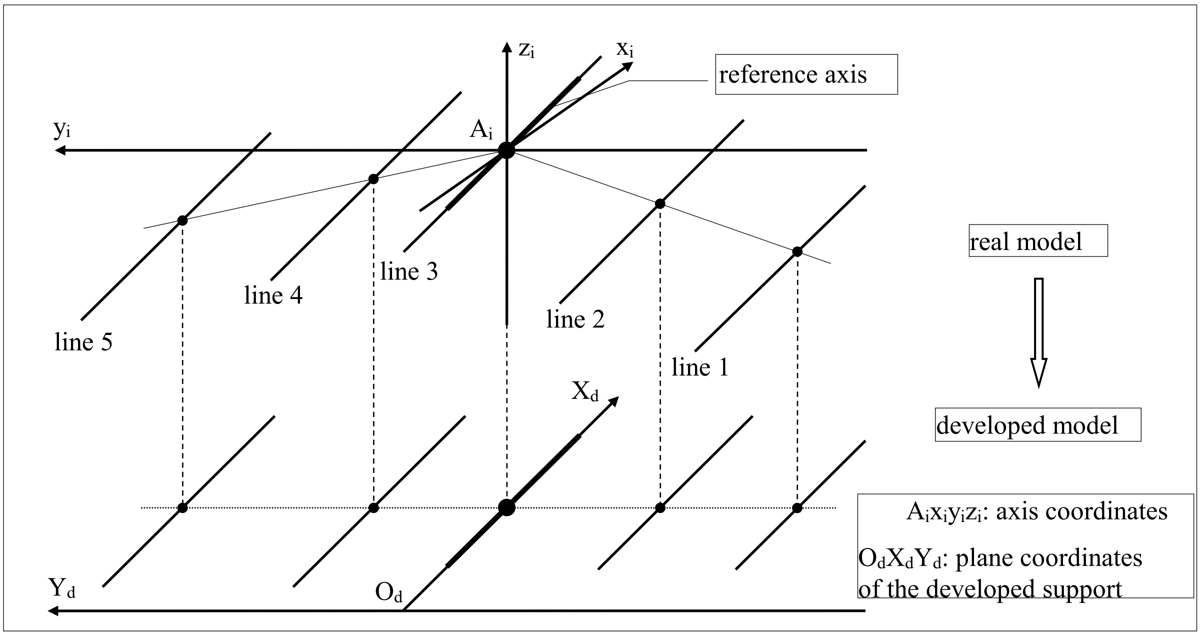


Figure 9.2 - Transverse correction of a loading surface

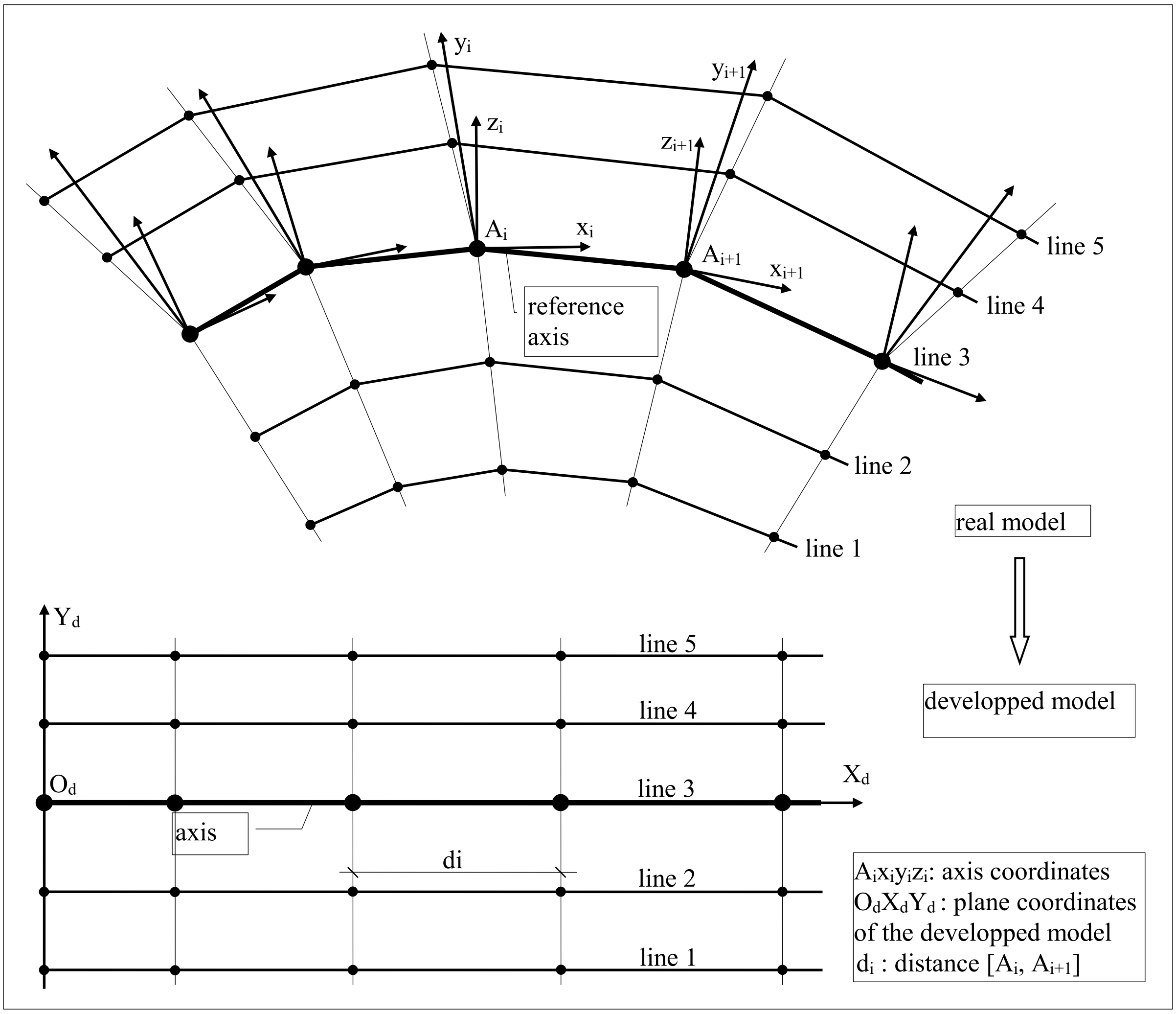


Figure 9.3 - Longitudinal development of a loading surface

The application of mobile loads to this developed model preserves the values of the surfaces of influence. Nevertheless, the limits of integration of the surfaces of influence are altered when changing from a non-flat loading surface to the equivalent developed and corrected surface, which amounts to a slight approximation.

Circulation

A circulation is a set of parameters defining the specific loading zones on a support (CHAUSSEES, TROTTOIRS), arranged transversally with regard to the axis.

The transverse footprint of a circulation may be inside the support, but it may also stretch beyond the support, because the two descriptions are independent. If the footprint stretches beyond the support, the corresponding influences are extrapolated.

Action

An action is a fixed or mobile loading system that acts on a support according to a single component. The distinction is made between:

- *predefined* regulatory actions that are identified by reserved names, with the circulations that must be associated with them completing the definition;
- *miscellaneous* actions that are entirely defined (nature, intensity and spacing of the loads, distribution zones, weighting coefficients, etc.), and are identified by newly created names.

Envelope effect

An envelope effect, or envelope, is the set of maximum and minimum values taken by the effect in question at each point of an area of study, and for a given component, when an action is applied to a support. Concomitant effects may be requested for components other than those being studied.

General conditions

The use of the ENV module is subject to the prior constitution of the general mechanical model by the PH1 module.

Areas of study, supports, circulations and actions can be defined, and then implicitly recorded, even if the database does not contain a structure saved by the PH3 module. The corresponding entities can be called by the current command file or, later on, by other command files.

The areas of study are used in most of the calculations made by the ETU module.

On the other hand, the calculation of a series of envelopes can only start if a saved loading structure is designated and called. This structure defines an activation system, the conditions at the limits and the rigidity, which the ENV module uses to determine the surfaces of influence.

Data analysis mode

The ENV module end delimiter commands can be written completely freely (from the point of view of their breakdown into lines), the standard labels integrated in their presentation are purely indicative.

They are analyzed in full, in the order in which they are introduced (interpretation). The ENV module may be asked to simply verify the commands, without executing them.

Editing

Commands are echoed as their interpretation proceeds; any erroneous commands are followed by error messages.

The results file always contains a recap of the entities introduced, the product of their pre-processing and the calculated envelope effects. The positions of the vehicles and certain intermediate results are produced as an option.

Contents

Command **Page**

9.1 - OVERLOADS

$$\text{SURCHARGES}\quad \left[\ {}^{*}\left\{\begin{array}{l}\text{VERIFIER}\\ \text{TITRE}\quad \text{titre\_session}\end{array}\right\}\right];$$

Parameters

- titre_session: title attributed to the command file that will be reproduced at the top of the results of the ENV module, if provided (string).

Functions

This command identifies an operational loads file and starts a "session" using the ENV module.

In VERIFICATION mode, the ENV module checks the syntax and logic of the commands, without recording the entities introduced, and without performing the calculations requested; the number of detectable errors is unlimited.

In EXECUTION mode (VERIFIER option not used), the introduced entities declared as correct are recorded. The validated calculation commands are executed and their results are recorded.

Conditions of use

- ❐ Must be at the beginning of the command file.

Methodological advice

- ❐ Always check the commands before starting an important calculation.

Examples

```
SURCHARGES
...........
FIN;
```

This SURCHARGES command label over two lines enables or disables the VERIFICATION mode, removing or restoring the "$" character of the first line.

```
SURCHARGES    $ CHECK
TITRE 'VIADUC D''ACCES B, CHARGES D''EXPLOITATION, SESSION 1';
..............
FIN;
```

Related commands

ETUDE; SUPPORT; CIRCULATION; ACTION; STRUCTURE; ENVELOPPES

RAPPEL; EFFACER; FIN

9.2 - STUDY

ETUDE no_domaine titre_domaine

$$\left\{\begin{array}{l} \text{DEPLACEMEN TS} \left\{\begin{array}{l} \text{NOEUDS} \quad \langle \text{no\_noeud} \rangle_{\text{nb\_noeuds}} \\ \text{EXTREMITES} \quad \left\langle [\left\{\begin{array}{c}+\\-\end{array}\right\}]\text{no\_elemen t} \right\rangle_{\text{nb\_extre}} \end{array}\right\} \\ \text{REACTIONS} \quad \text{NOEUDS} \quad \langle \text{no\_noeud} \rangle_{\text{nb\_noeuds}} \\ \left\{\begin{array}{l} \text{EFFORTS} \left\{\begin{array}{l}\text{ELEMENTS}\\ \text{SECTIONS}\end{array}\right\} \\ \text{CONTRAINTE S} \left\{\begin{array}{l}\text{NORMALES}\\ \text{TANGENTES}\end{array}\right\} \end{array}\right\} \text{EXTREMITES} \left\langle [\left\{\begin{array}{c}+\\-\end{array}\right\}]\text{no\_element} \right\rangle_{\text{nb\_extre}} \end{array}\right\} ;$$

Parameters

The numbers of nodes and ends of elements (respectively nb_noeuds and nb_extre) are implicitly defined by the numbers of corresponding values provided.

- no_domaine: identification number of the area of study to be created, positive and below 9999;
- titre_domaine: title that will be recalled at each use of the area of study, in the results of the ENV and ETU modules, and in the menus and drawings of the RES module (character string);
- no_noeud: number of a calculation node of the support REACTIONS or the DEPLACEMENTS;
- no_element: number of an element for the calculation of the DEPLACEMENTS, EFFORTS or CONTRAINTES, preceded by the *minus* sign, if the origin is designated, or the *plus* sign, or no sign, if the end is designated. With the DEPLACEMENTS option, this value implicitly designates the non-numbered node linked to the origin or the end of this element, by a rigid eccentricity element.

Functions

This command defines a numbered or explicit area of study that can be recorded in the database, if it is validated.

The different types of results that can be used to form areas of study, their numbered components and the corresponding abbreviations are described in Table 1.1.

Conditions of use

- The database can contain any number of areas of study.
- An area of study that is recorded during a session can be used in subsequent sessions.
- An area of study is replaced in the database when it is redefined, but the envelopes calculated with its old definition are only updated after the ENVELOPPES commands that use it have been executed again.

Methodological advice

- It is often necessary to study the results (forces and stress) on both sides of the sections of beams with cable stops, guys or supports.
- The time taken to calculate an envelope effect is proportional to the number of points in the corresponding area of study. This important criterion must be taken into consideration in the choice of the number of nodes in the areas of study.
- It is common practice to split a structure into several areas of study in order to reduce their size (a bridge deck comprising several spans that are taken separately, or in groups of two or three, for example).
- Displacements (DEPLACEMENTS) are always given in the global reference, whether the results are called with the node command (NOEUDS) or the extremity command (EXTREMITES). Displacements are always those of the nodes. For the other physical quantities (REACTIONS, forces (EFFORTS) and strains (CONTRAINTES)), the EXTREMITES option gives the results in the element reference, whereas the NOEUDS option give the results in the global reference.

Examples

```
ETUDE  10 'TABLIER NORD, DEPLACEMENTS POINTS D''ACCROCHAGE DES HAUBANS'
    DEPLACEMENTS EXTREMITES
     -707  -716  -725  -734  -743    -763  -772  -781  -790  -799
     -807  -816  -825  -834  -843    -863  -872  -881  -890  -899;

ETUDE  11 'TABLIER NORD, REACTIONS D''APPUIS'
    REACTIONS NOEUDS  1 36 86 121;

ETUDE  21 'TABLIER NORD, CONTRAINTES NORMALES TRAVEE 1'
    CONTRAINTES NORMALES EXTREMITES
    << (-I, I) > I = 1 A 35>;

ETUDE  32 'TABLIER NORD, CONTRAINTES TANGENTES TRAVEE 2'
    CONTRAINTES TANGENTES EXTREMITES
    << (-I, I) > I = 36 A 85>;
```

Related commands

SURCHARGES ; ENVELOPPES ; RAPPEL

9.3 - SUPPORT

SUPPORT no_support titre_support

$$\left\{\begin{array}{l} \text{SUPPORTS} \quad \left\langle \text{no\_support\_base} \right\rangle_{\text{nb\_support s}} \\ \text{POUTRES} \quad \left\langle \text{no\_poutre} \right\rangle_{\text{nb\_poutres}} \quad [\text{SECTIONS} \quad \text{s\_debut} \quad \text{s\_fin}] \\ \text{ELEMENTS} \quad \left\langle \text{no\_element} \right\rangle_{\text{nb\_elements}} \\ \text{NOEUDS} \quad \left\langle \text{no\_noeud} \right\rangle_{\text{nb\_noeuds}} \end{array}\right\}$$

$$[\text{AXE} \quad \left\{\begin{array}{l} \text{REFERENCE} \quad \text{ordre\_ref} \\ \left\{\begin{array}{l} \text{RELATIF} \\ \text{ABSOLU} \end{array}\right\} \quad \left\langle \text{x\_point} \quad \text{y\_point} \quad \text{z\_point} \quad \theta_x \quad \theta_y \quad \theta_z \right\rangle_{\text{nb\_points}} \end{array}\right\}]$$

$$[\text{FILES} \quad \text{YFILES} \quad \left\langle \text{y\_file} \right\rangle_{\text{nb\_files}} \quad [\left\{\begin{array}{l} \text{ZFILES} \quad \left\langle \text{z\_files} \right\rangle_{\text{nb\_files}} \\ \text{ZPOINTS} \quad \left\langle \left\langle \text{z\_point} \right\rangle_{\text{nb\_files}} \right\rangle_{\text{nb\_points}} \end{array}\right\}]] \;;$$

Parameters

The number of basic supports, beams, elements, nodes, lines and points of a line (respectively nb_supports, nb_poutres, nb_elements, nb_noeuds, nb_files and nb_points) are implicitly defined by the numbers of the corresponding values provided.

- no_support: identification number of the support to be created, positive and less than 90,000;
- titre_support: title that will be repeated, each time the support is used, in the results of the ENV module (character string);
- no_support_base: number of a predefined basic support, if the support to be created is made up of SUPPORTS. These supports must be joined *transversely* in a given order, so that the ordinates of their "merged" lines, expressed in the local reference frames of their axis reference frame, are strictly increasing. Their axes must have an equal number of points, but the abscissa of the points may be different from one support to another. A basic support cannot itself be made up of supports.
- no_poutre, s_debut, s_fin: number of a beam, number of a section of the beam no_poutre_1 marking the start of the support, number of a section of the beam $\text{no\_poutre}_{\text{nb\_poutres}}$ marking the end of the support, if the load-bearing structure is made up of POUTRES placed end to end in the common direction of travel.

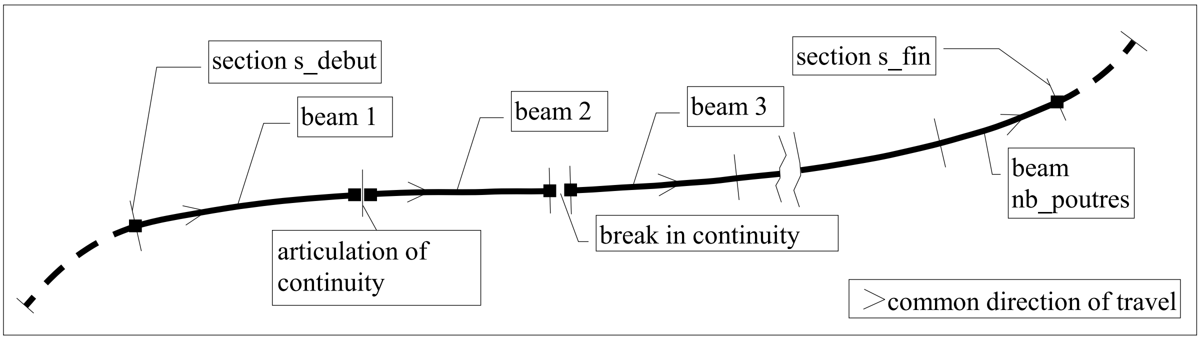


Figure 9.4 - Load-bearing structure made of beams

- no_element: number of a beam or common element, if the load-bearing structure is made of ELEMENTS;
- no_noeud: node number, if the load-bearing structure is made up of NOEUDS;
- ordre_ref: number of the order of introduction of the basic support, whose AXE is chosen as a REFERENCE, when the support to be created is made up of SUPPORTS. Therefore, the reference axis is the axis of the support $\text{no\_support\_base}_{\text{ordre\_ref.}}$
- x_point, y_point, z_point, θ_x, θ_y, θ_z: coordinates of a point A_i in the AXIS of the support to be created, and angles positioning its local reference frame $A_ix_iy_iz_i$, relative to a reference frame that depends on its mode of definition (AXE ABSOLU or RELATIF), and/or on the nature of the load-bearing structure (see general conventions and the analogy with the reference fiber of a beam in Chapter 3);
- y_file: ordinate of a line. The values entered must be strictly increasing;
- z_file: value of a line, when all the points of every line have the same value (ZFILES option);
- z_point: value of a point of a line, when the points of at least one line have different values (ZPOINTS option).

The values y_file, z_file and z_point are provided in the *local* reference frames of the axis reference frame of the support.

Functions

This command defines a load support that can be recorded in the database, if it is validated.

The support is described by three essential elements:

- the load-bearing structure of the ACTIONS;
- the reference AXE (which may also be used to define the CIRCULATION);
- the mode of discretization of the loading surface into FILES.

Load-bearing structure

It is introduced by the SUPPORTS, POUTRES, ELEMENTS or NOEUDS options and their attributes, which mean the following:

- SUPPORTS: the load-bearing structure is formed by *transversely* joining the structures of the nb_supports basic supports. The corresponding lines of nodes may be linked by transverse elements;
- POUTRES: the load-bearing structure is *longitudinally* made up of the mean fiber nodes of the nb_poutres beams, which are mechanically connected to one another. The first and/or the last beam may be partially taken into consideration;
- ELEMENTS: the load-bearing structure is *longitudinally* made up of the end nodes (taken together when contiguous) of the nb_elements elements, ordered according to their common direction of travel;
- NOEUDS: the load-bearing structure is *longitudinally* made up of the nb_nœuds nodes, ordered according to a given direction of travel.

When processing the ENVELOPPES command, the ENV module checks that all the elements of the load-bearing structures made up of beams or elements are active.

Two nodes less than 0.02 m apart only produce one discretization point of the reference axis (internal articulation in a beam, ends of elements or nodes that are close but not linked, etc.).

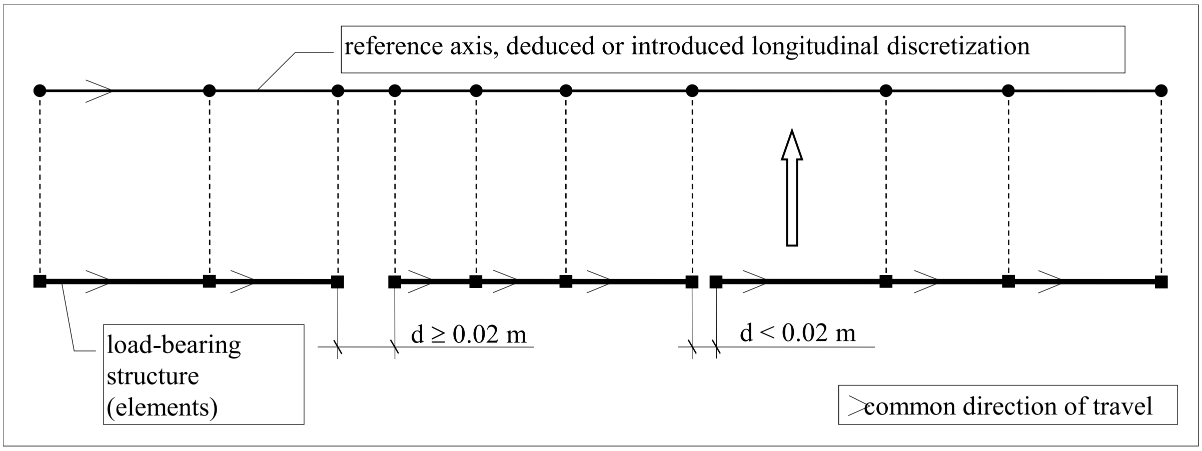


Figure 9.5 - Eliminating "merged" points in the reference axis

When processing the ENVELOPPES command, the ENV module checks that the elements declared to be load-bearing, or connected to nodes declared as load-bearing, are active in the structure being studied.

Reference axis

If the support is made up of supports, a reference axis must be designated (ordre_ref parameter).

If not, and if no axis is defined, it is merged with:

- the reference fibers placed end to end of the *beams* declared as load-bearing (possibly partial);
- the *end nodes* of elements, or the *nodes* declared to be load-bearing. The local reference frame of a point of the axis is then translated from the global reference frame to this point.

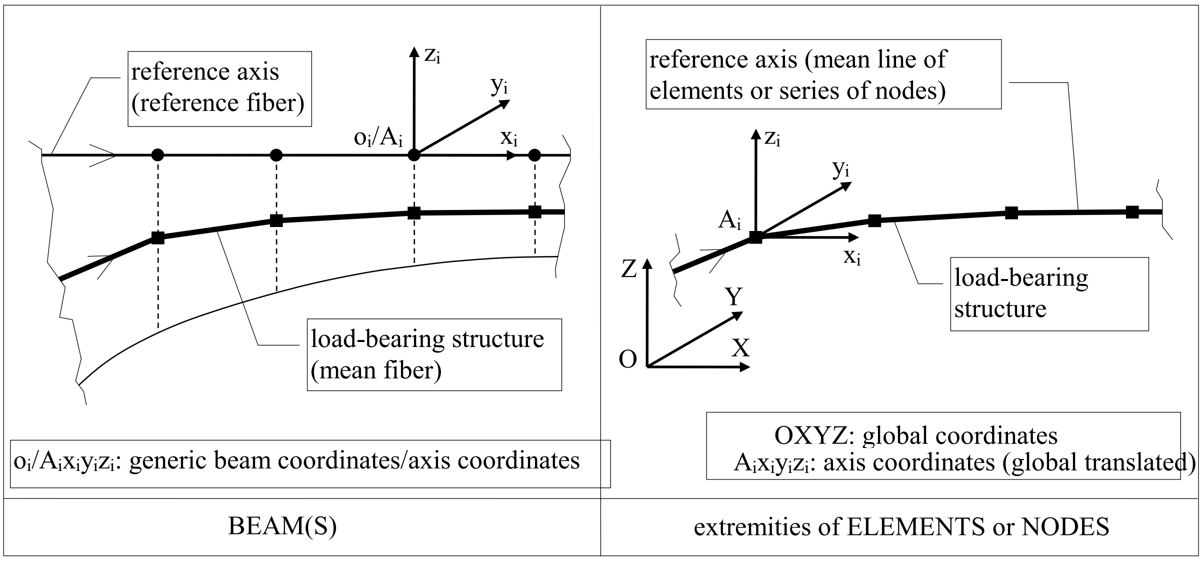


Figure 9.6 - Implicitly defined reference axis

If an axis is defined, the three coordinates of its points (corresponding to the longitudinal discretization of the load-bearing structure of the support), and the three angles positioning its local reference frames are defined:

- in the global reference frame, if the axis is declared as ABSOLU;
- if the axis is declared as RELATIF:
  - in relation to the points of the reference fibers of the beams declared as load-bearing, and in the corresponding local reference frame;
  - or in relation to points of the global reference frame translated to the end nodes of the elements, or to the nodes declared as load-bearing.

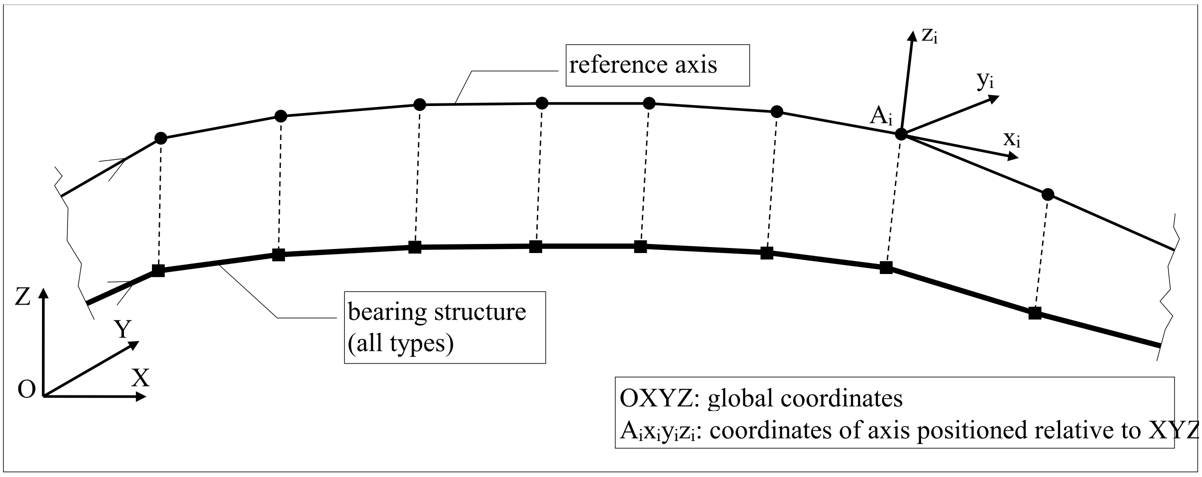


Figure 9.7 - Reference axis defined in an absolute reference frame

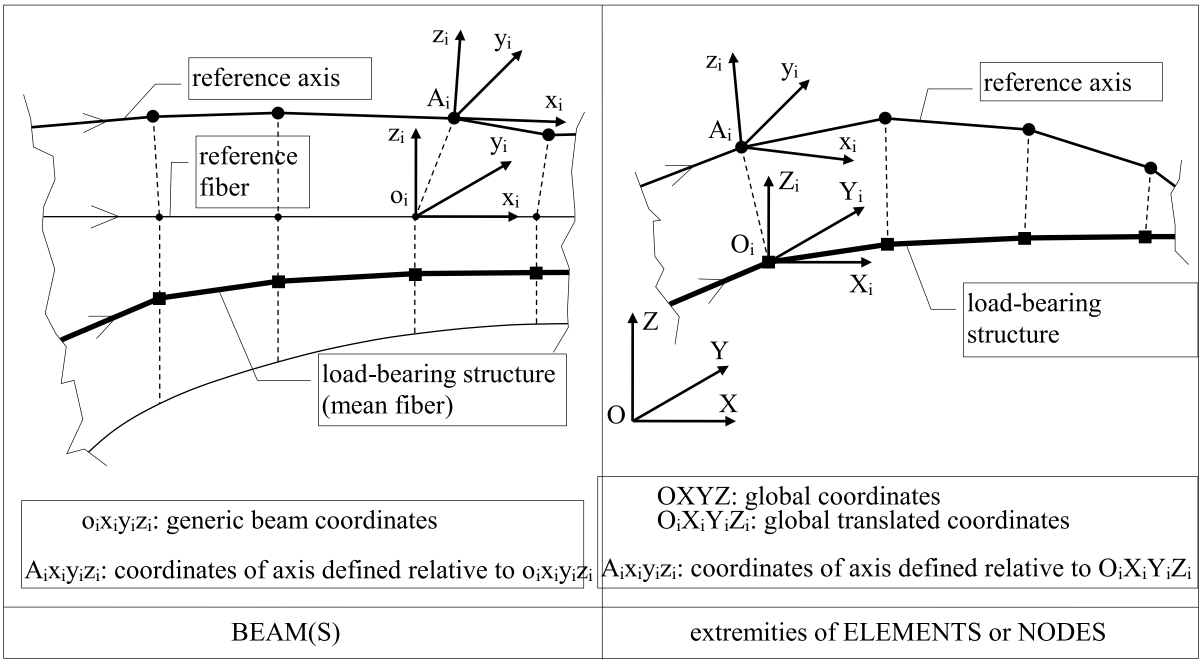


Figure 9.8 - Reference axis defined in relative reference frames

Lines

The lines determine the transverse topology of the support, i.e., the discretization lines of the surfaces of influence.

If the support is made up of supports, no lines must be defined. The lines of the constituent supports are used and merged in their order of introduction.

If not, and if no lines are defined, the support has a single line that matches its axis.

If lines are defined, their values may be zero (default option), constant (ZFILES option) or variable (ZPOINTS option).

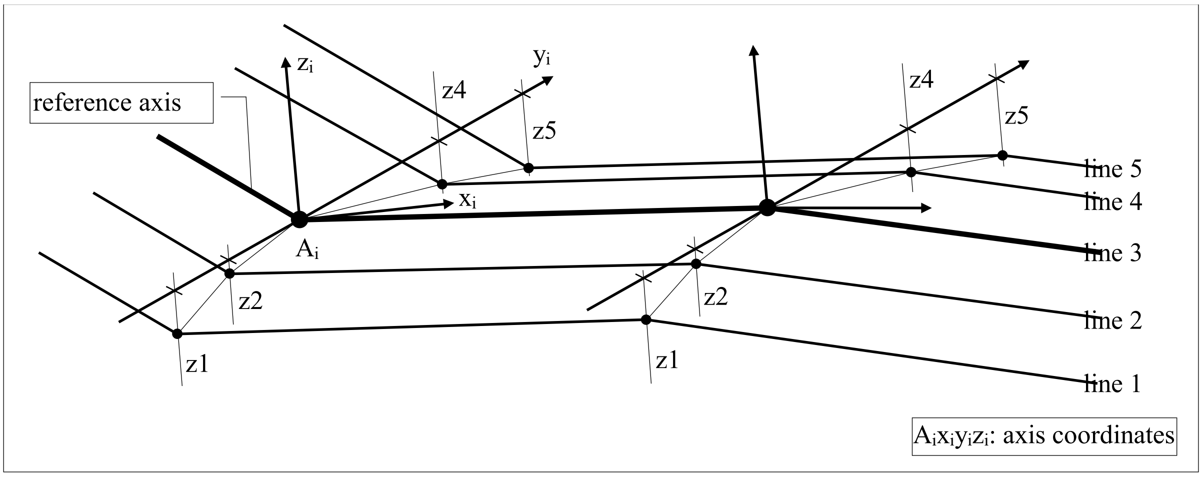


Figure 9.9 - Lines with longitudinally constant values

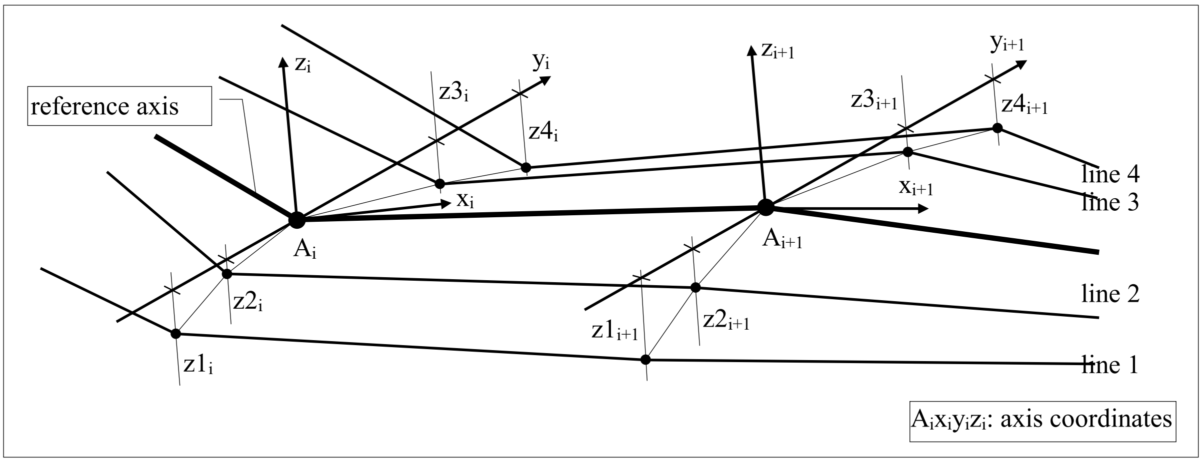


Figure 9.9 - Lines with longitudinally variable values

Conditions of use

- The database can contain any number of supports.
- A support that is recorded during a session can be used in subsequent sessions.

Methodological advice

- Supports made up of supports must always be used when a sub-structure exposed to mobile loads is modeled using parallel beams.
- Supports made up of nodes must be used to load nodes that are not connected by longitudinal elements that could constitute the load-bearing line (consideration of longitudinal discontinuities of the surfaces of influence).

- When modeling slabs linking two ribs by transverse elements, the intermediate nodes can be designated as support nodes.
- Match one line to the reference axis if it is an axis of a traffic lane or a pavement.
- Generally, no more than two lines correspond to each elementary support. These lines demarcate the limits of the surface of influence associated with the corresponding part of the structure. The surface of influence of a support made up of supports is interpolated between the end adjacent lines of the elementary supports.
- Skewed multi-beam constructions may be made up of several transversely concatenated supports, with each support having a different origin in order to take the skew into consideration. The only condition applying to the longitudinal distribution of the points is that the number of points of each support is the same from one support to another. The abscissa of each point can, therefore, be distributed differently from one support to another.

Examples

```
$ bearing structure made up of element extremities
$ reference axis (undefined) merged with their mean fiber
$ and with local coordinates of the translations of the global coordinates
$ only one strand (undefined) merged with the axis
SUPPORT  101  'TABLIER NORD'
    ELEMENTS  1 A 100;

$ bearing structure made up of two complete beams
$ placed end to end according to their common direction
$ reference axis (undefined) merged with their reference fibers
$ only one strand (undefined) merged with the axis
SUPPORT  102  'TABLIER NORD'
    POUTRES  1 2;

$ bearing structure made up of a part of a beam
$ bound by two of its sections
$ reference axis (undefined) merged with the portion of reference fiber
$ six strands with longitudinally constant dimensions
SUPPORT  103 'TABLIER NORD'
    POUTRE  1  SECTIONS  10  80
    FILES  YFILES  -9.00   -5.50   -2.00    2.00    5.50    9.00
           ZFILES  -0.2250 -0.1375 -0.0500 -0.0500 -0.1375 -0.2250;

$ bearing structure made up of element extremities
$ reference axis defined in global coordinates
$ two strands with longitudinally variable dimensions
SUPPORT  104 'TABLIER NORD'
    ELEMENTS  1 A 100
    AXE  ABSOLU
        << (X) 5*0.0 > X = 0.0 A 200.0 DIV 100 >
    FILES  YFILES  -2.0  2.0
           ZPOINTS << (Z,Z) > Z = 0.0 A 2.5 DIV 100 >;
```

The following example applies to a double-ribbed bridge deck, in which each rib is modeled by a beam and receives a support. A third support is constructed on the line of central intermediate nodes (not linked by longitudinal elements).

The resulting support is formed by transversely assembling these three basic supports, and the reference axis is chosen in the central slab. The ordinates of the seven resulting lines, which are recalculated in the reference frames of the points of this axis, are strictly increasing.

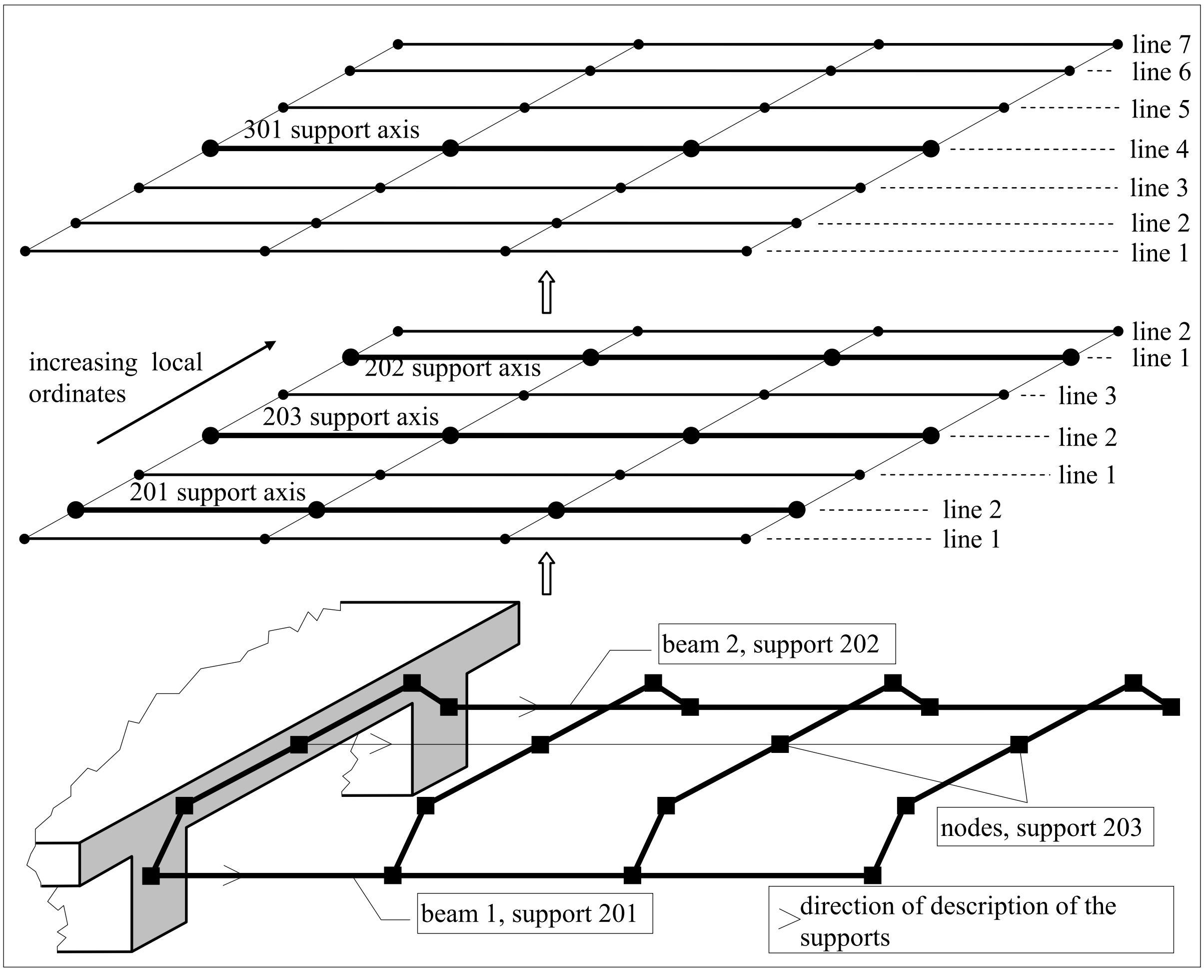


Figure 9.11 - Support of a double-ribbed bridge deck

```
$ one complete beam, reference axis: reference fiber, two strands
SUPPORT  201  'NERVURE DROITE' $ top view in the direction of the support
   POUTRE  1  FILES  YFILES  -1.0  0.0;

SUPPORT  202  'NERVURE GAUCHE'
   POUTRE  2  FILES  YFILES   0.0  1.0;

$ bearing structure made up of nodes, reference axis (undefined)
$ merged with the line of nodes, and with local coordinates that are
$ translations of the global coordinates, three strands
SUPPORT  203  'HOURDIS CENTRAL'
   NOEUDS  1 A 100  FILES  YFILES  -2.0  0.0  2.0;

$ resulting support. The reference axis is that of the support 203
$ with the order number 2 in the list of juxtapositions
SUPPORT  301  'TABLIER COMPLET'
   SUPPORTS  201  203  202  AXE  REFERENCE  2;
```

Related commands

SURCHARGES ; CIRCULATION ; ACTION ; STRUCTURE ; ENVELOPPES

RAPPEL

9.4 - CIRCULATION

CIRCULATION SUPPORT no_support

CLASSE classe_pont [<u>TYPE</u> $\left\{\begin{matrix}\underline{\textbf{RF61}}\\ \textbf{EC1}\end{matrix}\right\}$]

CHAUSSEES nb_chaussees $\langle \text{y\_debut} \quad \text{y\_fin} \rangle_{\text{nb\_chaussees}}$

[VOIES nb_voies largeur_voies $\langle \text{y\_axe\_voies} \rangle_{\text{nb\_voies}}$]

[TROTTOIRS nb_trottoirs $\langle \text{largeur\_trottoir} \quad \text{y\_axe\_trottoir} \rangle_{\text{nb\_trottoirs}}$] ;

Parameters

The TYPE RF61 options (selected by default) must be used with the predefined regulatory loads systems in Issue 61 or Circular 83, or the TYPE EC1 options with the systems in Eurocode 1 (see ACTION command).

- no_support: number of the support on which the circulation is defined;
- classe_pont: class of the bridge as per Issue 61 (1, 2 or 3), or of the circulation, as per Eurocode 1 (1, 2);
- nb_chaussees: number of carriageways, positive and limited to 2;
- y_debut, y_fin: strictly increasing ordinates of the limits of the *loadable* zone of a carriageway, defined by Issue 61, if the TYPE RF61 options are used, or demarcating its *effective* width, if the TYPE EC1 options are used;
- nb_voies: number of lanes;
- largeur_voies: width of the traffic lanes;
- y_axe_voies: ordinates of the axes of the lanes. The ordinates must be strictly increasing.
- nb_trottoirs: number of pavements, positive and limited to 2 if provided, zero by default;
- largeur_trottoir, y_axe_trottoir: width of a pavement and ordinate of its axis. The ordinates must be strictly increasing.

All the ordinates are provided in the *local* reference frames of the axis reference frame of the no_support support.

Functions

This command defines the circulation relative to a support that can be recorded in the database, if it is validated. The circulation of a support indicates its CLASSE and the geometry of its specific loading zones (CHAUSSEES, TROTTOIRS).

By default, the number of lanes and the width of the lanes in the carriageways are determined automatically according to the TRAFIC defined in the ACTION command, in both Eurocode and Issue 61.

When studying rail loads, the user must use the VOIES command to specify the positions and widths of the traffic lanes.

The software does not use the type of regulation. It is only requested here so that it appears in the data recap. The CLASSE is only used to determine the coefficients of degressivity to be applied to the TRAFIC, where appropriate, and to check the match with the CLASSE of the TRAFIC, where appropriate.

Conditions of use

- A circulation must be assigned to a support that receives predefined regulatory loads (see ACTION command).
- A support must be defined before the corresponding circulation, either in the same session or in an earlier session.
- A circulation that is recorded during a session can be used in subsequent sessions.
- A circulation is replaced in the database when it is redefined, but the envelopes calculated with its old definition are only updated after the ENVELOPPES commands that use it have been executed again (by the actions applied to a support).
- No formal rules link the definition of a circulation to that of its support.
- No overlap zones are tolerated between pavements and carriageways.
- In the case of two roads defined in the CIRCULATION, the potential loading model LM1 will include three lanes at the most, including one heavy lane.

Methodological advice

- When the type of circulation changes, check that the loadable width remains coherent with it.
- The fatigue loading model 3 (Fml3) is a model with a single vehicle. A single lane has to be define, the slow lane, with the VOIES command.
- In the particular case of two independent decks supported or not by the same piers, on which the user wants to apply one heavy lane and three lane at the most on each deck (see EC1-2 4.2.3(4)), two SUPPORTS must be defined with a CIRCULATION for each.
- In the particular case of two independent decks supported by the same piers, on which the user wants to apply one heavy lane and three lane at the most on the two decks for the study of the piers for example (see EC1-2 4.2.3(4)), only one SUPPORT must be defined with a CIRULATION including two roads.

Examples

The circulation represented below applies to a class one bridge with retaining elements only on the pavements. The central reservation cannot be removed. It can be linked to an analogy of the support 103, provided as an example of the SUPPORT command. It is studied according to Issue 61, then Eurocode 1.

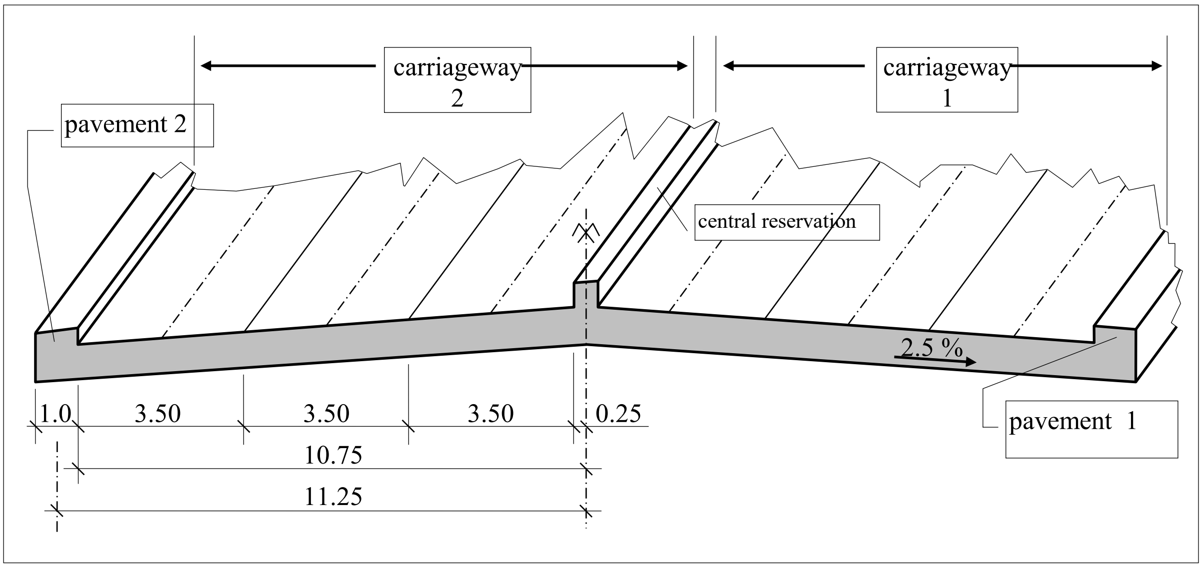


Figure 9.12 - Example of circulation

```
$ study according to Leaflet 61, width of the pavements reduced by 0.50
CIRCULATION  SUPPORT  103
    CLASSE  1  TYPE RF61
    CHAUSSEES  2  -10.25  -0.25    0.25  10.25
    TROTTOIRS  2    1.00 -11.25    1.00  11.25;

$ study according to Eurocode 1, width of the pavements not reduced
CIRCULATION  SUPPORT  103
    CLASSE  1  TYPE EC1
    CHAUSSEES  2  -10.75  -0.25    0.25  10.75
    TROTTOIRS  2    1.00 -11.25    1.00  11.25;
```

Related commands

SURCHARGES ; SUPPORT ; ACTION

9.5 - ACTION

ACTION no_action titre_action

SUPPORT no_support [COMPOSANTE no_comp]

$$\text{TRAFIC} \quad \left\{\begin{array}{l}\text{nom\_trafic\_predefini}\\ \text{nom\_trafic\_quelconque}\end{array}\right\}\left({}^{*}\left\{\begin{array}{l}\text{REMANENT}\\ \text{PONDERATION pond}\\ \text{BORNES} \qquad y_1 \quad y_2\end{array}\right\}\right) \; ;$$

Parameters

- no_action: identification number of the action to be created, positive and less than 90,000;
- titre_action: title that will be recalled at each use of the action, in the results of the ENV and ETU modules, and in the menus and drawings of the RES module (character string);
- no_support: number of the support to which the action is applied;
- no_comp: number of the acting components for the application of any loads (3 by default). See Introduction - Terminology - Support, and Figure 9.1;
- nom_trafic_predefini: name designating a system of predefined regulatory loads, chosen from the first column of tables 9.2, 9.3 or 9.4. The positive direction of circulation of the vehicles or convoys is that of the increasing abscissa, along the O_dX_d axis of the developed loading surface of the support (Figure 9.3);
- nom_trafic_quelconque: name designating any system of loads that is entirely defined.
- pond: weighting coefficient to be applied to the effect of the ACTION. This coefficient can be used to weight the effects of remanent actions before they are combined with other actions. For example, this coefficient corresponds to the global weighting coefficient of a special LM3-type vehicle - declared as remanent for LM1 - that combines the additional dynamic effect, the uncertainty coefficient and the coefficient ψ.
- y1, y2: transverse displacement boundaries of the traffic. These boundaries overload the LARGEUR_CHARGEABLE of the TRAFIC, irrespective of the parameters of the CIRCULATION command and the width of the lane for type_voie_5 TRAFIC (see the commands VOIES and LARGEUR_CHARGEABLE of TRAFIC). For special LM3-type vehicles, these boundaries are used to limit its transverse displacement in the CHAUSSEE of the CIRCULATION. This is the only relevant usage of these boundaries.

Functions

This command defines an action relative to a support that can be recorded in the database, if it is validated. The predefined regulatory load systems (reduced to their gravitational component) that are applicable to the carriageways and pavements can be invoked directly using their assigned and reserved names. For their complete description, refer to the documents mentioned in the Introduction.

For Issue 61, the reductive coefficients applicable to the load systems A and B are taken into consideration, but not the dynamic amplification coefficients applicable to the load system B.

Certain circulation parameters that are introduced subsequently may complete the definition of the predefined regulatory loads and specify where they are displaced. But the compatibility of the action and the circulation is only checked when the corresponding envelope effects are calculated.

With the REMANENT option, it is assumed that the traffic remains in position and **produces its effects** when the TRAFIC is subsequently processed in the ENVELOPPE command. This option overloads (modifies) the parameter defining the predefined type of MODALITE of the TRAFIC. It can only be applied to a TRAFIC that only comprises a CHARGE CONCENTREE. This option can be used to process special convoys traveling with the traffic. The option is applied to the TRAFIC defining the special convoy. In the ENVELOPPE command, the REMANENTE action must be followed by the corresponding TRAFIC LM1.

nom_trafic_predefini	Concise description
AL	Load A (l) *not limited by the floor value*
ALPL	Floor value of A (l)
TR	Uniform pavement load of 0.15 t/m^2
BCN, BCP	Truck B, traveling in the negative (BCN) or positive (BCP) direction
BT	Tandem B_t
C80_R, C80_C	M 80 military convoy, system M_c 80 simulated by three two-wheel axles with surface (MC80_R), or occasional (MC80_C) impacts
C120R, C120C	M 120 military convoy, system M_c 120 simulated by three two-wheel axles with surface (MC120_R), or occasional (MC120_C) impacts
CETD, CETE	Type D (CETD), or E (CETE)(1) special convoy

Table 9.2 - Predefined loads in Issue 61

nom_trafic_predefini	Concise description(1)
C1N, C1P	Special type C1 convoy traveling in the negative (C1N) or positive (C1P) direction
C2N, C2P	Special type C2 convoy traveling in the negative (C2N) or positive (C2P) direction
D2F1, D2F2, D3F1, D3F2	Type D.2F.1 (D2F1), D.2F.2 (D2F2), D.3F.1 (D3F1), or D.3F.2 (D3F2) special convoy
E2F1, E2F2, E3F1, E3F2	Type E.2F.1 (E2F1), E.2F.2 (E2F2), E.3F.1 (E3F1), or E.3F.2 (E3F2) special convoy

Table 9.3 - Predefined loads in Issue 83

nom_trafic_predefini	Concise description
E_M1C1_C, E_M1C2_C	Model 1, class 1 (E_M1C1_C), class 2 (E_M1C2_C), characteristic
E_M1C1_F, E_M1C2_F	As above for frequent loads
E_L1C1_C, E_L1C2_C	Model 1, class 1 (E_L1C1_C), class 2 (E_L1C2_C), characteristic for a local study or a traffic lane with a width of less than 3 m. The tandem occupies a width of 2.40 m.
E_L1C1_F, E_L1C2_F	As above for frequent loads for a local study or a traffic lane with a width of less than 3 m.
E_M2_C	Model 2, characteristic
E_B1C1_C, E_B1C2_C	Model 1, class 1 (E_B1C1_C), class 2 (E_B1C2_C), braking characteristic
E_FLM1	Fatigue load model 1
E_FLM2C1, E_FLM2C2, E_FLM2C3, E_FLM2C4, E_FLM2C5	Fatigue load model 2, Truck 1 (E_FLM2C1), Truck 2 (E_FLM2C2), Truck 3 (E_FLM2C3), etc.
E_FLM3	Fatigue load model 3
E_FLM4C1, E_FLM4C2, E_FLM4C3, E_FLM4C4, E_FLM4C5	Fatigue load model 4, Truck 1 (E_FLM4C1), Truck 2 (E_FLM4C2), Truck 3 (E_FLM4C3), etc.

Table 9.4 - Predefined road loads in Eurocode 1

nom_trafic_predefini	Concise description
LM4	Crowd load
TRR_EC	Uniform pavement load
PASSR_EC	Uniform footbridge load
TRC_EC	Concentrated pavement load
VSERV_EC	Service vehicle for footbridge/pavement

Table 9.5 - Predefined pedestrian loads in Eurocode 1

(1) Exceptional or military loads are defined in nominal values, without being assigned by weighting coefficients relating to flatness, the return period or dynamic effects. Therefore, these coefficients are to be defined by the user in the combinations of the ETU module or with the PONDERATION option of the command in progress, if it is an ACTION REMANENTE.

When convoys travel in traffic (TRAFIC REMANENT option), the effects of the special convoy and the effects of the normal traffic (frequent LM1) are aggregated by the ENV module for the effects of the normal traffic.

Miscellaneous traffic modules, defined using the additional commands of the TRAFIC submodule (see Chapter 10), are given new names that can be used to call them.

Conditions of use

- A predefined action can only be applied to a support that is already associated with a circulation.
- A support must be defined before the actions that are applied to it, either in the same session or in an earlier session.

- The database can contain any number of actions.
- An action that is recorded during a session can be used in subsequent sessions.
- An action is replaced in the database when it is redefined, but the envelopes calculated with its old definition are only updated after the ENVELOPPES commands that use it have been executed again.
- The identification number of an EFFET already recorded by the PH3, DYN or ETU modules (according to the terminology used by the ETU module, see Table 12.1) cannot be used to designate an ACTION.

Methodological advice

- Always check that the results of actions produced during the execution of the ENVELOPPES commands comply with the characteristics of the load systems to be produced.
- For tanks and special convoys, the amplitude of the transverse displacement of the loads can be reduced by modifying the width of the carriageway of the CIRCULATION command.

Examples

```
$ system of predefined regulatory highway loads
ACTION  100 'CHARGE A(l) SUR TABLIER NORD'
    SUPPORT  103  TRAFIC  AL;

$ system of any loads
ACTION  200 'CONVOI EXCEPTIONNEL NON REGLEMENTAIRE SUR TABLIER NORD'
    SUPPORT  103  TRAFIC  T_CONVU1;

$ excerpt of the commands of the TRAFIC submodule
$ introducing the exceptional convoy used above
DEBUT  T_CONVU1;
................
FIN;
```

Related commands

SURCHARGES ; SUPPORT ; CIRCULATION ; ENVELOPPES ; RAPPEL

9.6 - STRUCTURE

STRUCTURE nom_structure ;

Parameters

- nom_structure: name of a saved structure.

Functions

This command recalls a saved structure, on which the surfaces of influence and the envelope effects are calculated.

Conditions of use

- ☐ The called structure must already be recorded in the database, using the SAUVER command in the PH3 module (see Chapter 8).
- ☐ The saved structure must not provoke any instabilities (for example, due to the non-activation of articulations linking the parts of the structure). Its active elements must obey the rules applying to the definition of the supports that receive the actions invoked by the corresponding ENVELOPPES commands.
- ☐ A saved structure must be called before the envelope effects are calculated (see ENVELOPPES command), and its effect is remanent.
- ☐ Recalled structures can be replaced in the course of a same session.

Methodological advice

- ☐ For the ENV module, there is no difference between a structure that is saved at the end of construction or for an "infinite" time, if the static patterns are the same and if the Young's modulus of the materials have not varied.

Examples

These commands apply to the PH3 module, which produces two saved structures.

```
SAUVER  STRUCT_1
................
SAUVER  STRUCT_2
```

They are successively called by the ENV module to produce two "waves" of envelope effects.

```
STRUCTURE  STRUCT_1;
ENVELOPPES .......
ENVELOPPES .......
..................
STRUCTURE  STRUCT_2;
ENVELOPPES .......
..................
```

Related commands

SURCHARGES ; SUPPORT ; ENVELOPPES ; EFFACER

9.7 - ENVELOPES

$$\text{ENVELOPPES}\quad [\left\{\begin{array}{l}\text{SIMPLIFIEES}\\ \text{DEFAVORABLES}\end{array}\right\}]\quad [\left\{\begin{array}{l}\text{EDITER}\\ \text{INFLUENCES}\end{array}\right\}]$$

$$[\text{PRECISION}\quad \text{nb\_double\_x}\quad \text{nb\_double\_y}]$$

$$\left\langle\begin{array}{l}\left\{\begin{array}{l}\text{ETUDE no\_domaine}\\ \text{REACTIONS}\\ \text{EFFORTS SECTIONS [POUTRE no\_poutre]}\\ \text{DEPLACEMENTS}\\ \left\{\begin{array}{l}\text{EFFORTS ELEMENTS}\\ \text{CONTRAINTES}\left\{\begin{array}{l}\text{NORMALES}\\ \text{TANGENTES}\end{array}\right\}\end{array}\right\}\ [\left\{\begin{array}{l}\text{POUTRE no\_poutre}\\ \text{SECTION nom\_section}\\ \text{STANDARD}\end{array}\right\}]\\ \text{EFFORTS ELEMENTS}\ [\left\{\begin{array}{l}\text{BIARTICULATIONS}\\ \text{ARTICULATIONS}\end{array}\right\}]\end{array}\right\}\\ \quad [\left\langle\text{COMPOSANTE}\quad \text{no\_cpp}\quad [\text{CONCOMITANTES}\quad \langle\text{no\_cpc}\rangle_{\text{nb\_cpc}}]\right\rangle_{\text{nb\_cpp}}]\end{array}\right\rangle_{\text{nb\_domaines}}$$

$$\text{ACTIONS}\quad \langle\text{no\_action}\rangle_{\text{nb\_actions}}\ ;$$

Parameters

The number of numbered (explicit) or unnumbered (implicit) areas of study, nb_domaines, is implicitly defined by the number of calls to the ETUDE option or to its concurrent options at the same level.

The numbers of principal components, concomitant components to a principal component and actions (respectively nb_cpp, nb_cpc and nb_actions) are implicitly defined by the numbers of corresponding values provided.

- nb_double_x, nb_double_y: (positive) numbers of duplicates of the calculation points of the surfaces of influence, along the axes O_dX_d and O_dY_d of the developed loading surface of the support, coupled with the actions (Figure 9.3), when the precision is increased. By default, these values are zero and the initial surfaces are preserved;
- no_domaine: number of the area of study, not of the DEPLACEMENTS type, for which the envelope effects are calculated. If the ETUDE option is not used, the study applies to a non-numbered area;
- no_poutre: the beam number for which EFFORTS are studied in SECTIONS or ELEMENTS reference frames, or the CONTRAINTES;
- nom_section: name of a section-type, defined by a SECTION TYPE command of the PH1 module (see Chapter 6); all elements having received this section-type assignment are implicitly designated for the study of EFFORTS in ELEMENTS or CONTRAINTES reference frames;
- no_cpp, no_cpc: number of a principal study component, and an associated concomitant component, to be chosen according to the conventions of Table 1.1;
- no_action: number of an action to be applied.

Functions

This command prompts the calculation of the envelope effects for certain explicit or implicit areas of study, and certain possible main and concomitant components, produced by a series of actions, applied by their reference support to a saved structure.

To perform these calculations, the ENV module determines the surfaces of influence corresponding to the areas of study and to the designated components, and records them in the database. They can then be reused for other actions applying to the same components of the same areas of study.

The envelope effects recorded in the database can be read by the ETU module, reproduced and combined with other envelope effects, or with other effects produced by the PH3, DYN or ETU modules.

If no study component is specified for an area, all its components are studied in order, without any concomitant values. The study of concomitant components must always be explicitly requested.

With the SIMPLIFIEES option, all the declared vehicle wheels with non-zero impact surfaces are assimilated with concentrated loads.

With the DEFAVORABLES option, the unfavorable part of the impact surfaces of the vehicle wheels is taken into consideration, and the favorable part is ignored.

The EDITER option produces the intermediate results and the vehicle positions. By default, only the envelope effects are produced.

With the INFLUENCES option, the surfaces of influence are produced and recorded in the database in a format that allows the RES module to read and display them.

With the REACTIONS option, all the support reactions are produced.

With the STANDARD, BIARTICULATIONS or ARTICULATIONS options, all standard elements (current not bi-articulated), bi-articulations or articulations are considered.

In the absence of further additional options, the following are considered: all beam elements for the study of forces in section reference frames, all elements for the study of forces in element reference frames, or all beam (and standard) elements for the study of stresses.

The PRECISION option increases the number of points of the surfaces of influence.

When ACTION REMANENTE is used, the order of the ACTIONS in the ENVELOPPE command is important. An ACTION REMANENTE must always be followed by an ACTION, to which the REMANENCE is applied. The REMANENCE is cumulative, inasmuch as all the consecutive ACTIONS REMANENTES preceding an ACTION are REMANENTES for this action. For example, if the actions 1, 2 and 3 are remanent, and 4, 5 and 6 are not remanent in the command:

```
ENVELOPPE
ACTIONS 1 2 3 4 5 6;
```

Then action 1 is REMANENTE for 2, actions 1 and 2 are REMANENTES for 3 and actions 1, 2 and 3 are REMANENTES for 4. Actions 5 and 6 are not affected by any remanence. In other terms, the effect of action 2 is the aggregate of the effects of actions 1 and 2. The effect of action 3 is the aggregate of the effects of actions 1, 2 and 3. The effect of action 4 is the aggregate of the effects of actions 1, 2, 3 and 4.

Conditions of use

- Any calculation of envelope effects must be preceded by a call to a saved structure (see STRUCTURE command).
- All the actions invoked in an ENVELOPPES command must refer to the same support.
- A series of envelope effects is replaced in the database when the corresponding ENVELOPPES command is re-executed (for example, after modifying certain actions that it invokes).
- The last specified ACTION cannot be REMANENT.

Methodological advice

- When the areas of study only differ in terms of their types and concern the same nodes or element ends, the time taken to calculate their corresponding envelope effects can be shortened by grouping them in a single ENVELOPPES command, or by processing them in a series of ENVELOPPES commands, without inserting any EFFACER INFLUENCES commands.
- The calculation of envelope effects is done in a linear way. If the user wants a non-linear calculation, he could find the most unfavourable position of the traffic load for the studied section thanks to the surface of influence. This loading case should be defined in the phase module PH3 by creating equivalent loads. The results of these loading cases will take into account the non-linear effect.

Examples

In the example below, the first area of study, unnumbered, is a "support reaction", the area of study 21 is a "forces in section reference frames", and the area of study 31 is "normal stress". The 12 distinct components produced are as follows:

- for the first area (studied in full), six main reaction components (RFX, RFY, RFZ, RMX, RMY and RMZ);
- for area 21, one main component (FSX), and four concomitant force components (FSY, FSZ, MSX and MSY);
- for area 31, one main stress component (SIG).

```
$ the three actions invoked must be applied to the same support
ENVELOPPES  DEFAVORABLES  EDITER
    REACTIONS
    ETUDE  21    COMPOSANTE  1    CONCOMITANTES  2 3 4 5
    ETUDE  31
    ACTIONS  100 200 300;
```

In the example below, which applies to the same areas of study, the ten following distinct components are generated:

- for the first area, three main reaction components (RFZ, RMX, RMY);
- for area 21, six main components (FSX, FSY, FSZ, MSX, MSY and MSZ), and six concomitant force components (the same ones);
- for area 31, one main stress component (SIG).

```
ENVELOPPES
    REACTIONS    COMPOSANTE  3    COMPOSANTE  4    COMPOSANTE  5
    ETUDE  21    COMPOSANTE  1    CONCOMITANTES  2 3 4 5 6
```

```
                COMPOSANTE  2    CONCOMITANTES  1 3 4 5 6
                COMPOSANTE  3    CONCOMITANTES  1 2 4 5 6
                COMPOSANTE  4    CONCOMITANTES  1 2 3 5 6
                COMPOSANTE  5    CONCOMITANTES  1 2 3 4 6
                COMPOSANTE  6    CONCOMITANTES  1 2 3 4 5
   ETUDE  31
   ACTIONS  100 200 300;
```

The example below can be used to produce the surfaces of influence, calculated with greater precision (twice and four-fold), and to record them in the database so that they can be viewed in the RES module. The envelopes are not calculated.

```
ENVELOPPES  INFLUENCES  PRECISION 1 2  ETUDE 21  ACTIONS  100;
```

Related commands

SURCHARGES ; ETUDE ; SUPPORT ; ACTION ; STRUCTURE ; EFFACER

9.8 - RECALL

$$\text{RAPPEL} \quad [\text{TERMINAL}] \quad {}^{*}\left\{\begin{matrix}\text{SAUVEGARDE S}\\ \text{ETUDES}\\ \text{SUPPORTS}\\ \text{ACTIONS}\end{matrix}\right\} ;$$

Functions

This command sends the following entities in the database to the screen (TERMINAL option) or to a results file (default option):

- the names of the structures saved by the PH3 module (SAUVEGARDES option);
- the numbers and the characteristics of the areas of study (ETUDES option), the supports (SUPPORTS option), and/or the actions (ACTIONS option), recorded by the ENV module.

Examples

```
$ SCREEN display of the names of the structures saved by module PH3
RAPPEL  TERMINAL  SAUVEGARDES;

$ save to FILE the results of the numbers and the characteristics
$ of all the entities defined and saved by the ENV module
RAPPEL  ETUDES SUPPORTS ACTIONS;
```

Related commands

SURCHARGES ; ETUDE ; SUPPORT ; ACTION

9.9 - DELETE

EFFACER INFLUENCES ;

Functions

This command deletes all the surfaces of influence recorded in the database that are related to a recorded structure. Their space is made available to record other data.

Conditions of use

- Any deletion of surfaces of influence must be preceded by a call to a saved structure (see the command STRUCTURE).

Methodological advice

- It is advisable to use this command at the end of each session, during which surfaces of influence are calculated.
- If the size of the mechanical model and/or the size of the areas of study result in high volumes of stored data, the EFFACER command can be used after each ENVELOPPES command (see also the methodological advice for the ENVELOPPES command).
- The physical space that is freed by the deletion can only be used by other files if the database has been compressed.

Examples

Consider a bridge deck with three spans, each of which constitutes the basis of one of the nine following areas of study:

  - areas 11, 21 and 31: forces in section reference frames, normal stress, tangential and normal stress, in span 1, respectively:
  - areas 12, 22 and 32: same in span 2;
  - areas 13, 23 and 33: same in span 3.

The envelope effects for two applied actions are calculated in three successive waves. If the areas of study of a span cover the same element ends, the calculation time will be shortened by grouping them together.

```
STRUCTURE  STRUSERV  $ structure saved at the called commissioning
ENVELOPPES    ETUDE  11  ETUDE  21  ETUDE  31    ACTIONS  100 200;
EFFACER  INFLUENCES;
ENVELOPPES    ETUDE  12  ETUDE  22  ETUDE  32    ACTIONS  100 200;
EFFACER  INFLUENCES;
ENVELOPPES    ETUDE  13  ETUDE  23  ETUDE  33    ACTIONS  100 200;
EFFACER  INFLUENCES;
FIN;
```

Related commands

SURCHARGES ; STRUCTURE ; ENVELOPPES

9.10 - END

FIN;

Functions

This command causes the end of a session and stops the execution of the ENV module; all potential following commands are ignored.

Examples

```
SURCHARGES
TITRE  ’TABLIER NORD, EFFORTS ET CONTRAINTES SOUS CHARGES D’’EXPLOITATION’;
..............
..............
FIN;
```

Related commands

SURCHARGES

Chapter 10

Definition of traffic

Introduction

Annex A provides the full wording of the documents referenced in this chapter under the condensed names: "Issue 61" and "Eurocode 1".

Annex C contains a comprehensive typical example annotated in the form of a "skeleton" and actual complete examples of the use of the TRAFIC submodule.

General concepts

The CHARGES managed by the TRAFIC submodule are applied to a flat and rectangular surface called the ZONE CHARGEABLE, which is longitudinally divided up into rectangular strips or VOIES, with a BANDE CENTRALE that can be closed off.

The longitudinal and transverse footprint of the loadable zone, and the central reservation, can be explicitly defined, or defined by referring to certain commands of the ENV module that call on defined traffics.

Lanes

Lanes are an essential concept in the data model of the TRAFIC submodule. A lane is a surface inside which a *load* can be applied. Lanes can be defined explicitly, or by referring to certain parameters of the CIRCULATION command in the ENV module.

The TRAFIC submodule offers several modes to split the loadable zone into lanes. In fact, this is the submodule's fundamental mechanism:

- the positions and the widths of the lanes can be defined in advance (e.g., pavements);
- the loadable zone can be broken down into contiguous lanes according to precise rules (Issue 61 or UK regulations, for example);
- several lanes can be moved transversely, by fixing the first lane in its least favorable position, then the second lane in the same way, in the rest of the loadable zone, and so on (Eurocode 1);
- finally, several lanes can be moved transversely, while considering the least favorable combination of lanes.

The *degressivity coefficients* weight the effects of the loads applied to the lanes, whenever they are provided. They are given as a function of the number of lanes, and can be linked to *classes* of *circulation*.

Loads

A load is *unfavorable* when its *maximum* effect produces a *positive* result, and its *minimum* effect produces a *negative* result. A load is *favorable* when its *maximum* effect produces a *negative* result, and its *minimum* effect produces a *positive* result.

A *load applicable to a lane* can be made up of two types of loads that are not mutually exclusive:

- the CHARGE UNIFORME is applied to the surfaces of influence in their entirety, or only to their unfavorable parts. The intensity of this load can vary depending on the loaded length and according to one of the proposed laws of variation. The effect of a uniform load may be weighted differently, depending on whether it is favorable or unfavorable. Three cases may be considered:
  - when looking for the least favorable *combination of areas* (like for the A loads system in Issue 61);
  - when *combining all the unfavorable areas* (like for the uniform load in Eurocode 1);
  - when *combining all the areas*, irrespective of their signs.
- the CHARGE CONCENTRÉE is a CONVOI of an unlimited number of one or more identical or different VÉHICULES. Each vehicle has a certain number of ESSIEUX and each axle is made up of any number of ROUES. A wheel may have a *point* or *surface* impact on the loadable zone. The favorable effect of each vehicle, axle or wheel may be weighted. The effect of a concentrated load may be weighted differently, depending on whether it is favorable or unfavorable.

When a lane can *simultaneously* receive a uniform load and a concentrated load, the loads may be considered to be *independent* or *linked*, in one of the two following ways:

- the uniform load is *removed* in the *removal zones* attached to the concentrated load, defined relative to the front and the rear of each vehicle;
- the intensity of the concentrated load depends on the corresponding intensity of the uniform load and, therefore, on the loaded length.

Modalities, traffic and remanence

The loading MODALITES include the limits of the loadable zone (*length* and *width*) and the manner in which it is split into lanes, while automatically taking a possible central reservation into consideration. TRAFIC comprises a list of loads applicable to each of the lanes, according to the predefined modalities.

The total orthogonality between the possibilities of lane loading and their transverse processing allows all the possible combinations of loads to be simulated.

A modality is REMANENTE if the vehicles in the traffic that it defines remain in place and produce their effects for the following traffic, in the order in which the traffics are defined in the ENVELOPPE command. In this case, the vehicles in the following traffic cannot occupy the removal zones of the remanent vehicles and the uniform loads of the following traffic are removed in the removal zone of the remanent vehicles. By extension, a MODALITE REMANENTE defines a TRAFIC REMANENT and the VEHICULES REMANENTS. Remanence only applies to VEHICULES. There is no remanence of CHARGES UNIFORMES.

The figure below shows all the types of entities that the TRAFIC submodule can manage, with a representation of their main logical links. This diagram is read from top to bottom.

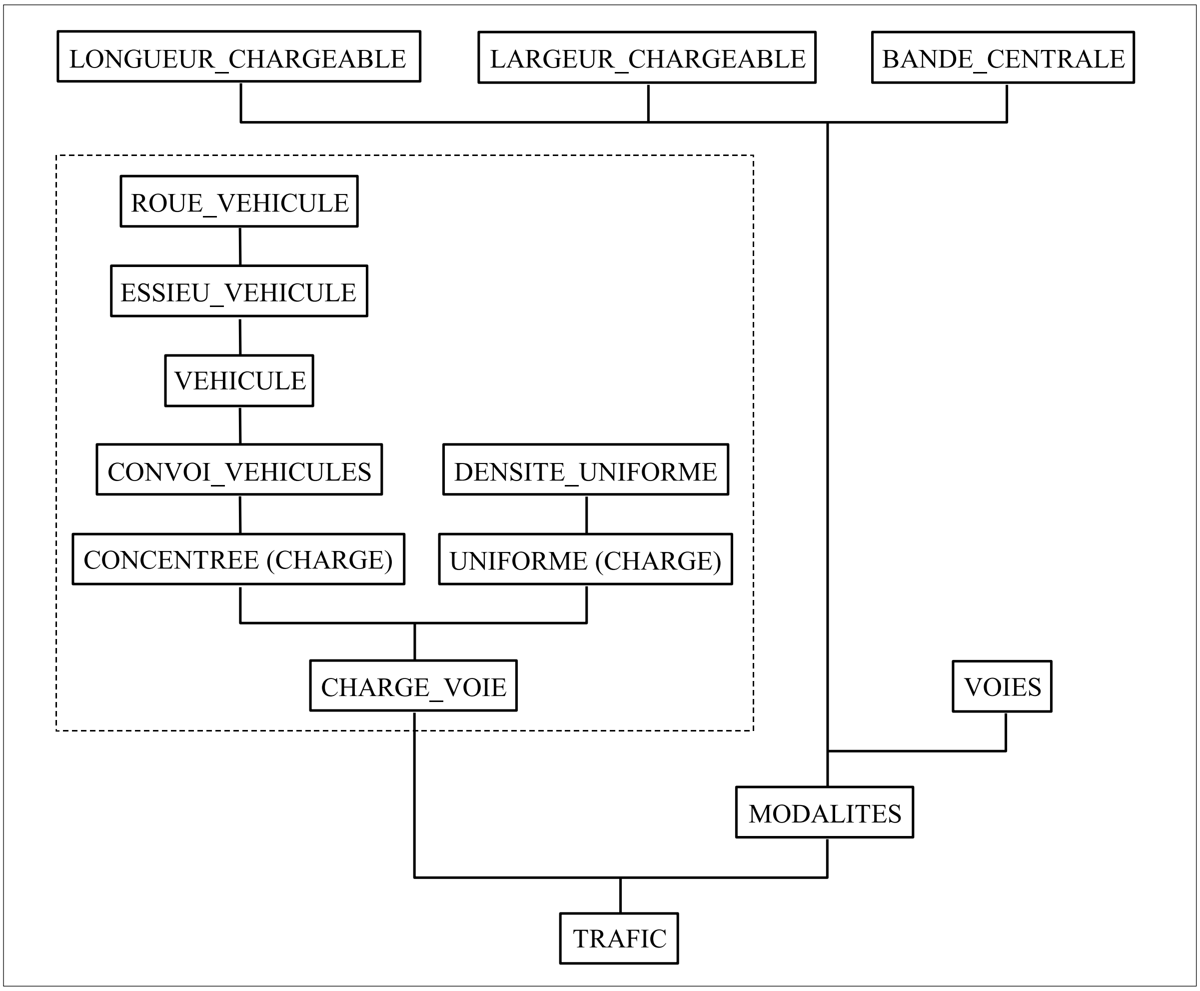


Figure 10.1 - Hierarchy of the concepts (or commands) of the TRAFIC submodule

Surfaces of influence

Surfaces of influence are made up of lines of points in parallel vertical planes. For the ENV module, the abscissa of lines are concordant and the lines have the same number of points. The TRAFIC submodule can process lines that do not display these particularities.

The longitudinal and transverse axes of a surface of influence are the same as the $O_dX_dY_d$ reference frame of the developed loading surface of the *support* of the ENV module, on which it is built (Figure 9.3).

As a general rule, the higher the number of points defining the surfaces of influence, the more precise the calculations. Some of their details can be correctly modeled by locally adding definition points.

When a single line of points is defined, the resulting surface of influence, which does not vary transversely, is recognized and analyzed using simplified algorithms that avoid any unnecessary transverse processing.

Commands

The various defined traffics must be introduced sequentially, and in any order, into the *trafic.don* file dedicated to a project.

The TRAFIC submodule end delimiter commands can be written completely freely (from the point of view of their breakdown into lines), the standard labels integrated in their presentation are purely indicative.

They are analyzed in full, in the order in which they are introduced (interpretation). Then, their content is partially compiled.

Identification numbers

For simplicity's sake, the *identification numbers*, sometimes referred to as *numbers*, are integers (positive and less than 1,000) that are used to designate the various *entities* making up a TRAFIC data model, and are assigned when they are created. They are not visible beyond the traffic in which they appear.

A number that is already used to designate an entity cannot be reused to designate another entity, irrespective of the type, in the same traffic.

An entity can be made up of other entities that are identified by their numbers. For example, a set of several wheel entities forms an axle entity.

A zero must be used when a compulsory parameter must not refer to any particular entity. This entity must always be optional.

In the order of a flow of commands, references "in advance" to entities that have not yet been created are often possible, but only in the same traffic. However, all the commands that can only call on entities that have already been created are associated with a suitable comment.

Predefined constants

TRAFIC data contains *constants* that identify *types of entities*. For example, the type of the lanes is encoded by an integer with a value of between 0 and 6.

To facilitate the creation and the interpretation of the command files, these constants are *predefined* and can be called as *variables* of the PCP language using their reserved names *in brackets* and separated by commas, if there are more than one. For example: (type_voies_5), (bande_1, bande_n_char), etc.

These variables are used instead of the corresponding whole constants in the titles presenting the commands or the sections describing their functions. The list of these named constants and their values is provided at the end of this chapter.

External references

When the description of a command refers to a support, this support is used by the ACTION commands in the ENV module that call the traffic. Certain data that may be attached to the circulation can be reused by the TRAFIC submodule.

Contents

10.1 - START

DEBUT nom_trafic ;

Parameters

- nom_trafic: the name given to a traffic, used to call it in an ACTION command in the ENV module. It must start with the characters "T_".

Functions

This command is used to start the definition of a traffic and to identify it using a name.

Conditions of use

- ☐ Must be at the start of each group of commands defining a traffic.

Examples

```
$ start of file trafic.don containing two traffics
DEBUT  T_CONVU1;
$ definition of the type U1 convoy
................
................
FIN;

DEBUT  T_CONVU2;
$ definition of the type U2 convoy
................
................
FIN;
```

Related commands

FIN

10.2 – CLASSE_CIBLE

CLASSE_CIBLE   classe ;

Parameters

- classe: number of the target class of the TRAFIC

Functions

Identifies the target class of the TRAFIC. This command is optional, because certain TRAFICS have multiple classes.

10.3 - LONGUEUR_CHARGEABLE

$$\text{LONGUEUR\_CHARGEABLE} \quad \text{no\_longueur} \quad \begin{Bmatrix} \text{SUPPORT} & \\ \text{x\_zone\_inf} & \text{x\_zone\_sup} \end{Bmatrix} ;$$

Parameters

- no_longueur: identification number assigned to the loadable length.
- x_zone_inf, x_zone_sup: abscissa of the lower and upper limits of the loadable zone defined in the longitudinal axis O_dX_d of the surfaces of influence. x_zone_sup must be greater than x_zone_inf.

Functions

This command defines the longitudinal footprint of the loadable zone relative to that of the surfaces of influence of a support. A wheel located outside the longitudinal footprint of the surfaces of influence, or outside the loadable zone is not taken into account in the global calculation of the effect of a vehicle in motion.

If the SUPPORT option is used, the longitudinal footprint of the loadable zone is that of the developed loading surface of the support.

If x_zone_inf and x_zone_sup are defined (Figure 10.2):

- if the loadable zone only covers a part of the surfaces of influence, all their points located outside the loadable zone are assumed to have zero influence.
- if the loadable zone stops short of and/or beyond the limits of the surfaces of influence, the parts of the loadable zone located longitudinally outside the surfaces of influence are assumed to have zero effect.

Conditions of use

- A traffic must only contain a single command of this type, inserted before the MODALITES command that refers to it.
- When its limits are explicitly defined, the loadable zone can be positioned anywhere longitudinally, but it must cover at least a part of the surfaces of influence.

Examples

```
$ loadable zones longitudinally merged
$ with the developed loading surface of a support
LONGUEUR_CHARGEABLE  (longueur_1)  SUPPORT;
................
MODALITES  (..., ..., ..., longueur_1, ..., ..., ...);

$ longitudinal footprint of the loadable zone explicitly defined
LONGUEUR_CHARGEABLE  (longueur_1)  -1.0 125.0;
```

Related commands

MODALITES

10.4 - LARGEUR_CHARGEABLE

$$\text{LARGEUR\_CHARGEABLE} \quad \text{no\_largeur} \quad \left\{ \begin{array}{l} \text{CIRCULATION} \quad \left\{ \begin{array}{l} \text{CHAUSSEES} \\ \text{TROTTOIRS} \end{array} \right\} \\ \text{y\_zone\_inf} \quad \text{y\_zone\_sup} \end{array} \right\} ;$$

Parameters

- no_largeur: identification number assigned to the loadable width.
- y_zone_inf, y_zone_sup: ordinates of the lower and upper limits of the loadable zone defined in the transverse axis O_dY_d of the surfaces of influence. y_zone_sup must be greater than y_zone_inf.

Functions

This command defines the transverse footprint of the loadable zone relative to that of the surfaces of influence of a support. A wheel located outside the longitudinal footprint of the surfaces of influence, or outside the loadable zone is not taken into account in the global calculation of the effect of a vehicle in motion.

The transverse footprint of the loadable zone is that of:

- the circulation carriageways, including a possible central reservation, if CIRCULATION CHAUSSEES is encoded,
- the circulation, including the pavements, if CIRCULATION TROTTOIRS is encoded.

If y_zone_inf and y_zone_sup are defined (Figure 10.2):

- if the loadable zone only covers a part of the surfaces of influence, all their points located outside the loadable zone are assumed to have zero influence.
- if the loadable zone stops short of and/or beyond the limits of the surfaces of influence, the parts of the loadable zone located transversely outside the surfaces of influence are *transversely extrapolated*.

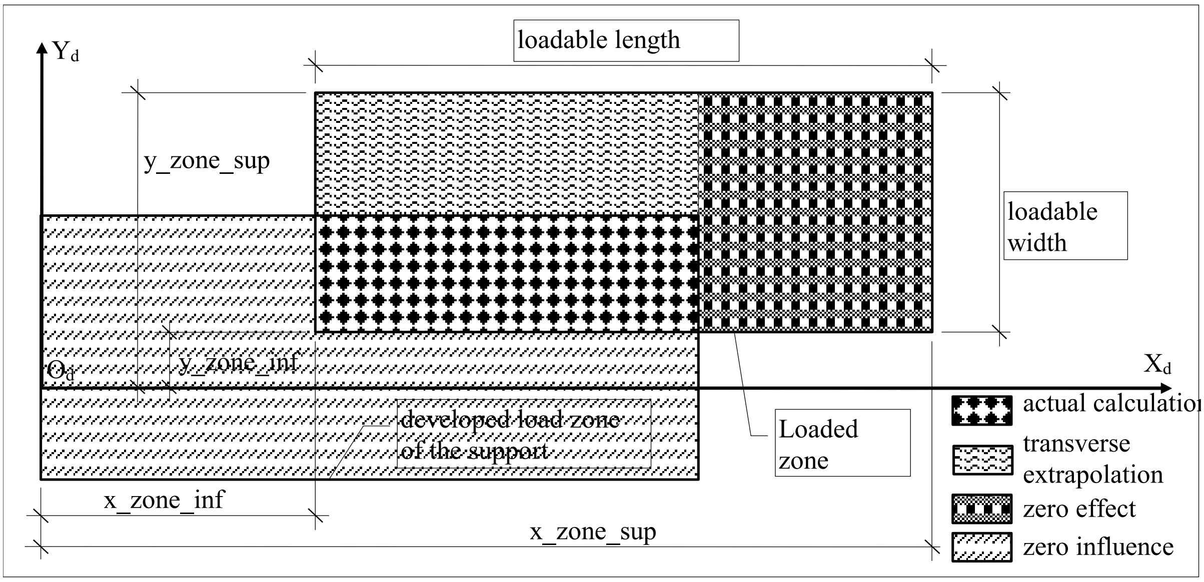


Figure 10.2 - Support and loadable zone (rectangles with parallel sides)

Conditions of use

- A traffic must only contain a single command of this type, inserted before the MODALITES command that refers to it.
- When its limits are explicitly defined, the loadable zone can be positioned anywhere transversely, relative to the surfaces of influence.

Examples

```
$ loadable zone transversely merged with the traffic pavements
LARGEUR_CHARGEABLE  (largeur_1)  CIRCULATION CHAUSSEES;
................
MODALITES  (..., ..., largeur_1, ..., ..., ..., ...);

$ transverse footprint of the loadable zone explicitly defined
LARGEUR_CHARGEABLE  (largeur_1)  -10.75 10.75;
```

Related commands

MODALITES

10.5 - BANDE_CENTRALE

$$\text{BANDE\_CENTRALE} \quad \text{no\_bande} \quad \text{type\_bande} \quad \begin{Bmatrix} \text{CIRCULATION} \\ \text{y\_bande\_inf} \quad \text{y\_bande\_sup} \end{Bmatrix} ;$$

Parameters

- no_bande: identification number assigned to the central reservation.
- type_bande: type of central reservation. Can take one of the following values:
  - (bande_n_char): indicates that the central reservation is not loadable and cannot be covered by lanes;
  - (bande_char): indicates that the central reservation is loadable and can be covered by lanes, but the part of the surfaces of influence it covers is set to zero;
- y_bande_inf, y_bande_sup: ordinates of the lower and upper limits of the central reservation defined in the transverse axis O_dY_d of the surfaces of influence. y_zone_sup must be greater than y_zone_inf. and the central reservation must be inside the transverse footprint of the loadable zone.

Functions

This command defines the transverse footprint of the central reservation relative to that of the surfaces of influence of a support.

If the CIRCULATION option is used, and if the circulation has two carriageways, the central reservation is the zone that separates them. If there is only one carriageway, then there is no central reservation.

Irrespective of its type, the separation of the loadable zone into lanes (VOIES command) is:

- *influenced* by a *loadable* central reservation, that no lanes must cover, even partially;
- *not influenced* by a central reservation that is *not loadable*.

Conditions of use

- ☐ A traffic can only contain a single command of this type, inserted before the MODALITES command that refers to it.

Examples

```
$ non-loadable central reservation between the traffic pavements
BANDE_CENTRALE  (bande_1, bande_n_char)  CIRCULATION;
................
MODALITES  (..., ..., ..., ..., ..., ..., bande_1);

$ transverse footprint of the loadable central reservation defined
$ explicitly
BANDE_CENTRALE  (bande_1, bande_char)  -0.25 0.25;
```

Related commands

MODALITES

10.6 - ROUE_VEHICULE

ROUE_VEHICULE no_roue type_roue impact_x impact_y
type_longueur lg_min lg_imposee_max lg_imposee_min
coef_a ponder_f_roue nb_val $[\langle \mathrm{L} \quad \mathrm{Q} \rangle_{\mathrm{nb\_val}}]$;

Parameters

- no_roue: identification number assigned to the wheel;
- type_roue: type of variation of the intensity Q of the weight of the wheel, as a function of the conventional loaded length L. Can take one of the following values:
  - (roue_tabl): intensity defined by a list of couples of values $\langle \mathrm{L} \quad \mathrm{q} \rangle_{\mathrm{nb\_val}}$. The current intensity is calculated by linear interpolation in the domain covered by the list, or using the value of the nearest boundary, outside the domain of the list:

$$\begin{aligned} &Q = Q_i + (Q_{i+1} - Q_i)\big[(L - L_i)/(L_{i+1} - L_i)\big] && \text{pour : } L_i < L \le L_{i+1} \\ &Q = Q_1 && \text{pour : } L \le L_1 \\ &Q = Q_{nb\_val} && \text{pour : } L > L_{nb\_val} \end{aligned} \qquad (10.1)$$

  - (roue_cons) :

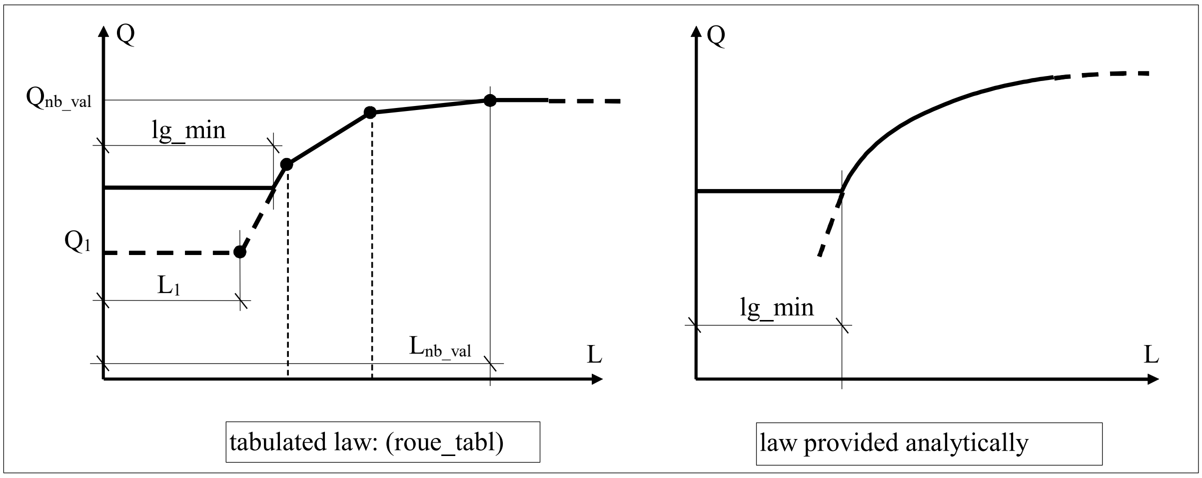
$$Q = a \qquad (10.2)$$

  - (roue_raci) :

$$Q = a\sqrt{L} \qquad (10.3)$$

  - (roue_loga) :

$$Q = a \ln(L) \qquad (10.4)$$



tabulated law: (roue_tabl)

law provided analytically

Figure 10.3 - Types of laws of variation for Q(L)

- impact_x, impact_y: the longitudinal and transverse dimensions of the wheel's impact surface on the loading zone, assumed to be rectangular with sides parallel to the axes of the $O_dX_dY_d$ reference frame of the surfaces of influence. These values must both be positive for surface impact (Figure 10.4), or both zero for point impact.

- type_longueur: type of determination of the *conventional* loaded lengths (named as such because they can be calculated or imposed) used to calculate the intensity of the weight of the wheel. Can take one of the following values:
    - (longueur_calc): indicates that the conventional loaded lengths are those of the zones located between the points at which the lines of influence pass through zero. These lines are determined when the concomitant uniform load is applied to the lane in question (which must then exist in the CHARGE_VOIE command).
    - (longueur_impo): indicates that the conventional loaded lengths are fixed by the parameters lg_imposee_max and lg_imposee_min, depending on whether we are looking for the maximum or the minimum effect of the wheel.
- lg_min: the loaded length below which the intensity of the weight of the wheel is supposed to be constant (Figure 10.3). This minimum length is used, irrespective of the value of the type_longueur parameter.
- lg_imposee_max, lg_imposee_min: loaded length used to calculate the intensity Q in order to study the maximum and minimum effects respectively, if the wheel is of the imposed length type (the value of the type_longueur parameter is (longueur_impo)).
- coef_a: coefficient that may be used in the formulas 10.2 to 10.4.
- ponder_f_roue: weighting coefficient to be applied to the effect of the wheel, only if it is favorable. Must be positive or zero.
- nb_val: the number of couples of values (L, Q) to be provided. Greater than one if the intensity of the weight of the wheel is of the type (roue_tabl), otherwise zero.
- L, Q: conventional loaded length and intensity of the weight of the wheel defining a point of discretization of the tabulated variation law Q(L). The values of L must be strictly increasing.

Functions

This command defines a vehicle wheel by its impact surface and the intensity of its weight, which may be constant, or may depend on the conventional loaded length of the concomitant uniform load, or by parameters that are provided directly. This conventional loaded length may be the result of an effective calculation or may be imposed.

A wheel located outside the longitudinal footprint of the surfaces of influence, or outside the loadable zone is not taken into account in the global calculation of the effect of a vehicle in motion.

The mode of integration of the surfaces of influence in the impact zone of a wheel is determined by the option SIMPLIFIEES or DEFAVORABLES or, by default, by the ENVELOPPES command in the ENV module. If the impact of the wheel is a point impact, these choices do not affect the calculations. Otherwise, the integration may apply to the entire impact surface, or only the portion of the surface that has an unfavorable effect.

The favorable weighting coefficient applies to the effects of the wheel, independently of the effects of the other wheels.

The unit used corresponds to the unit defined in PH3 (by default, kdaN). The traffic loads introduced by the user must be defined in the unit chosen in the phasing.

Conditions of use

- ☐ A traffic may contain any number of this type of commands.

Examples

```
$ can apply to the predefined exceptional convoy C1N/C1P of the ENV module
$ three types of wheels defined: two with surface impact, one with
$ occasional impact, the intensity of their weight Q is not dependent on a
$ conventional loaded length, favorable effect weighting coefficient of
$ 1.00
$ parameters common to the three types of wheels defined in one go
<lg_min           = 0.00>
<lg_imposee_max   = 1.00>
<lg_imposee_min   = 1.00>
<ponder_f_roue    = 1.00>
<nb_val           = 0>
$ -----------------------
<impact_x         = 0.37>
<impact_y         = 0.37>
<coef_a           = (6.00/2.00)>
ROUE_VEHICULE  (roue_1, roue_cons, impact_x, impact_y,
               longueur_impo, lg_min, lg_imposee_max, lg_imposee_min,
               coef_a, ponder_f_roue, nb_val);
<impact_x  = 0.37>
<impact_y  = (0.37+0.37)>
<coef_a    = (12.00/2.00)>
ROUE_VEHICULE  (roue_2, roue_cons, impact_x, impact_y,
               longueur_impo, lg_min, lg_imposee_max, lg_imposee_min,
               coef_a, ponder_f_roue, nb_val);
<impact_x  = 0.00>
<impact_y  = 0.00>
<coef_a    = (12.80/4.00)>
ROUE_VEHICULE  (roue_3, roue_cons, impact_x, impact_y,
               longueur_impo, lg_min, lg_imposee_max, lg_imposee_min,
               coef_a, ponder_f_roue, nb_val);
<nb_roues = 2>
ESSIEU_VEHICULE  (..., ..., nb_roues) (nb_roues)*(roue_1) (..., ...);
................
```

Related commands

UNIFORME ; ESSIEU_VEHICULE

10.7 - ESSIEU_VEHICULE

ESSIEU_VEHICULE no_essieu ponder_f_essi

nb_roues $\langle \text{no\_roue} \rangle_{\text{nb\_roues}}$ $\langle \text{y\_roue} \rangle_{\text{nb\_roues}}$;

Parameters

- no_essieu: identification number assigned to the axle.
- ponder_f_essi: weighting coefficient to be applied to the effect of the axle, only if it is favorable. Must be positive or zero.
- nb_roues: number of wheels making up the axle. Must be positive.
- no_roue: number of a wheel; defined on a ROUE_VEHICULE command.
- y_roue: ordinate of the longitudinal axis of a wheel, defined in the $O_vx_vy_v$, reference frame, specific to the vehicle to which the axle belongs, which is a translation of the $O_dX_dY_d$ reference frame of the surfaces of influence. The origin of this reference frame must be on the lower limit of the transverse footprint of the vehicle, while its longitudinal position is indifferent. The ordinates of the wheels must be strictly increasing.

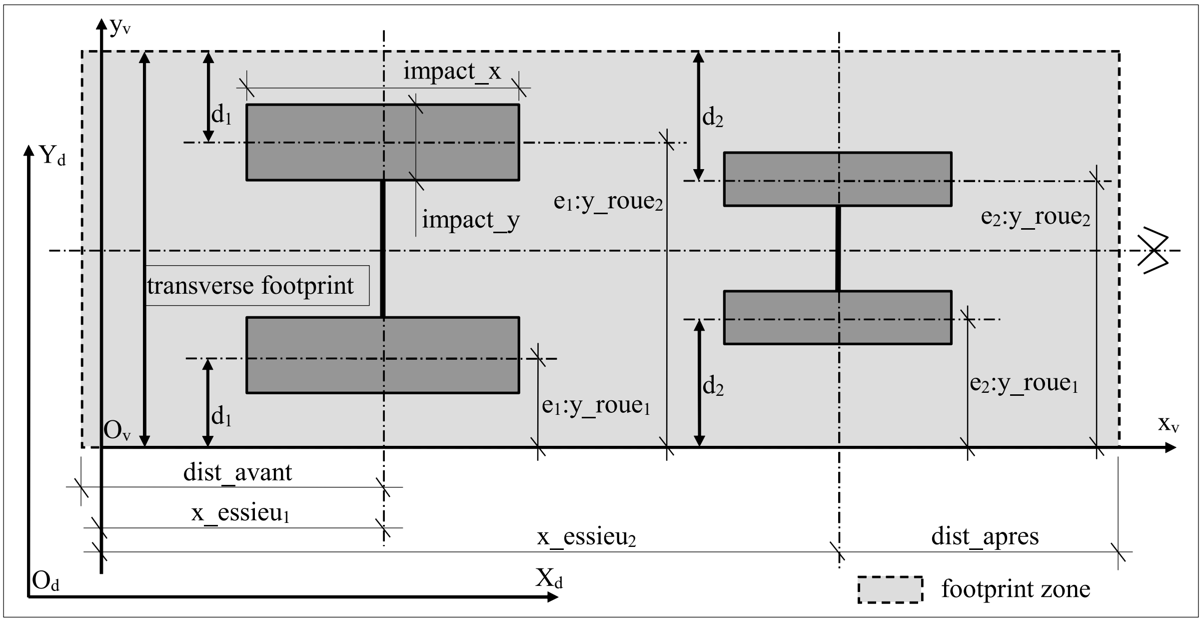


Figure 10.4 - Vehicle with two two-wheel axles with surface impacts

Functions

This command defines an axle made of wheels transversely positioned in a reference frame attached to a vehicle. The transverse footprint of the vehicle, which is assumed to be transversely symmetrical, is determined by adding the ordinate of the first wheel, to the center-to-center distance of the end wheels on each side of the axle.

A wheel located outside the longitudinal footprint of the surfaces of influence, or outside the loadable zone, is not taken into account in the global calculation of the effect of a vehicle in motion. The favorable weighting coefficient applies to the effects of the axle, independently of the effects of the other axles.

Conditions of use

- A traffic may contain any number of this type of commands.
- Several vehicles with at least one axle in common must have the same transverse footprint.
- The transverse footprint of a vehicle must not exceed the width of a lane.

Examples

```
$ can apply to the predefined load BCN/BCP of the ENV module that
$ contains two types of axles, each with two identical wheels with
$ an occasional impact, favorable effect weighting coefficient of 1.00
$ parameters common the the two types of axles defined in one go
................
<nb_val = 0>
ROUE_VEHICULE   (roue_1, ..., ..., ..., ..., ...,
                ..., ..., ..., ..., nb_val);
................
ROUE_VEHICULE   (roue_2, ..., ..., ..., ..., ...,
                ..., ..., ..., ..., nb_val);
<ponder_f_essi= 1.00>
<nb_roues     = 2>
<y_roue1      = 0.25>
<y_roue2      = 2.25>
ESSIEU_VEHICULE   (essieu_1, ponder_f_essi, nb_roues)
                  (nb_roues)*(roue_1) (y_roue1, y_roue2);
ESSIEU_VEHICULE   (essieu_2, ponder_f_essi, nb_roues)
                  (nb_roues)*(roue_2) (y_roue1, y_roue2);
```

Related commands

ROUE_VEHICULE ; VEHICULE

10.8 - VEHICLE

VEHICULE no_vehicule ponder_f_vehi type_enleve_unif
dist_avant dist_apres enleve_avant enleve_apres
nb_essieux $\langle \text{no\_essieu} \rangle_{\text{nb\_essieux}}$ $\langle \text{x\_essieu} \rangle_{\text{nb\_essieux}}$;

Parameters

- no_vehicule: identification number assigned to the vehicle.
- ponder_f_vehi: weighting coefficient to be applied to the effect of the vehicle, only if it is favorable. Must be positive or zero.
- type_enleve_unif: type of removal of the concomitant uniform load. Only used if the type_enleve_unif parameter of the CHARGE_VOIE command is (enlever_unif). Can take one of the following values:
  - (laisser_unif), (enlever_unif): respectively indicate that the concomitant uniform load (or future, in the case of remanence) is left or removed, in the removal zone of the vehicle's uniform load.
- dist_avant, dist_apres: distance before the axis of the first axle and distance after the axis of the last axle, longitudinally demarcating the vehicle's footprint zone, that no footprint of another vehicle can cover, even partially, unless the type_concentree parameter of the CONCENTREE command is (vehicule_born) (Figures 10.4 and 10.5).
- enleve_avant, enleve_apres: distance before the axis of the first axle and distance after the axis of the last axle, demarcating the removal zone of any concomitant uniform load, and future in the case of remanence.
- nb_essieux: number of axles making up the vehicle. Must be positive.
- no_essieu: number of an axle. Defined on an ESSIEU_VEHICULE command.
- x_essieu: abscissa of the axis of an axle, defined in the $O_vx_vy_v$ reference frame (see ESSIEU_VEHICULE command, y_roue parameter and Figure 10.4). The abscissa of the axles must be strictly increasing.

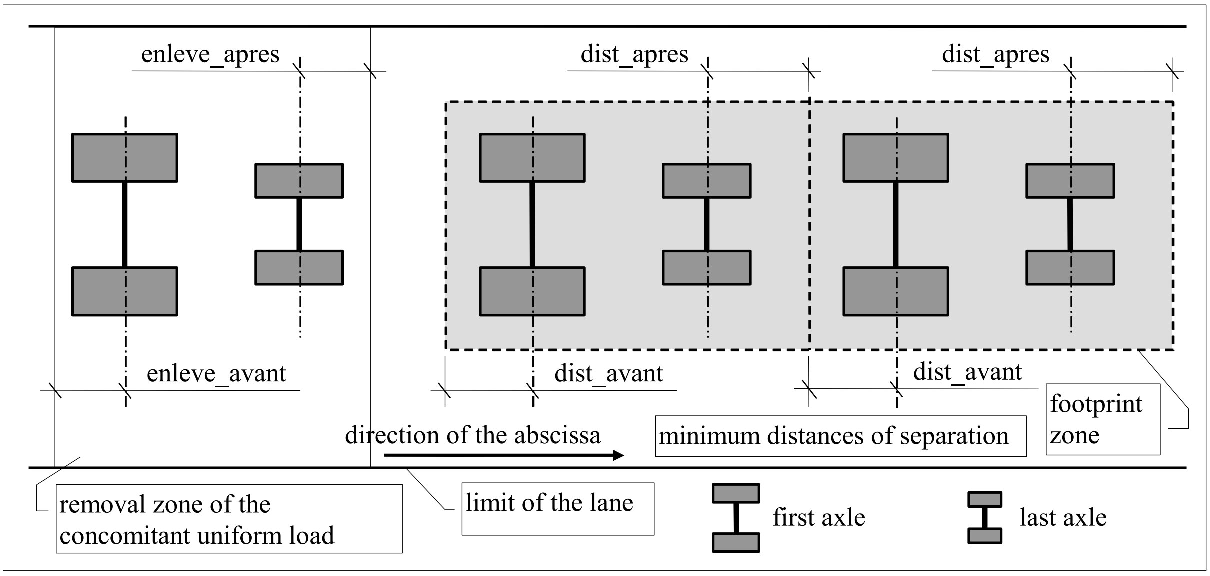


Figure 10.5 - Vehicles

Functions

This command defines a vehicle made up of axles that are longitudinally positioned in a reference frame attached to it. The transverse footprint of the vehicle is that of its axles. Its longitudinal footprint is obtained by adding the values provided to the center-to-center distance of its end axles. The longitudinal footprint of the removal zone of the concomitant uniform load can be defined in the same manner, independently of the footprint zone of the vehicle.

The removal distance is used for the uniform load of the current TRAFIC, if the option type_enleve_unif of the VEHICULE and of the CHARGE is enlev_unif.

CAUTION: for current REMANENT TRAFIC (see MODALITE):

- The removal zone is also used for the uniform loads of the following TRAFICS if the current option of the VEHICULE type_enleve_unif is enlev_unif, irrespective of the type_enleve_unif option of the current CHARGE and of the CHARGES of the following TRAFICS.
- **The removal zone is also used as a distance between vehicles for the VEHICULES of the following TRAFICS, irrespective of the type_enleve_unif option of the current CHARGE and of the CHARGES of the following TRAFICS.**

A wheel located outside the longitudinal footprint of the surfaces of influence, or outside the loadable zone, is not taken into account in the global calculation of the effect of a vehicle in motion.

The favorable weighting coefficient applies to the effects of the vehicle, independently of the effects of the other vehicles.

Conditions of use

- A traffic may contain any number of this type of commands.

Methodological advice

- A vehicle traveling in the "negative" direction can be obtained by longitudinally turning round its counterpart traveling in the "positive" direction.
- A vehicle traveling in the positive direction has its first referenced axle at the rear and its last referenced axle at the front. The *dist_avant* parameter is measured relative to the *rearmost* axle and the *dist_apres* parameter is measured relative to the *forwardmost* axle.

Examples

```
$ can apply to the predefined exceptional convoy C1P of the ENV module
$ vehicle with eight axles of three different types, longitudinal
$ footprint merged with the wheelbase of the extreme axles (vehicle
$ travelling alone), no concomitant uniform load removal zone
$ defined, favorable effect weighting coefficient of 0.0
<nb_roues = 2>
ESSIEU_VEHICULE  (essieu_1, ..., nb_roues)
                  ..., ..., ..., ...;
<nb_roues = 2>
ESSIEU_VEHICULE  (essieu_2, ..., nb_roues)
                  ..., ..., ..., ...;
<nb_roues = 4>
ESSIEU_VEHICULE  (essieu_3, ..., nb_roues)
                  ..., ..., ..., ...,   ..., ..., ..., ...;
<ponder_f_vehi= 0.00>
<dist_avant   = 0.0>
<dist_apres   = 0.0>
<enleve_avant = 0.0>
<enleve_apres = 0.0>
<nb_essieux   = 8>
<x_essieu1    = 0.00>
<x_essieu2    = 2.75>
<x_essieu3    = 4.10>
<x_essieu4    = 9.10>
<x_essieu5    = (9.10+1.0*1.55)>
<x_essieu6    = (9.10+2.0*1.55)>
<x_essieu7    = (9.10+3.0*1.55)>
<x_essieu8    = (9.10+4.0*1.55)>
VEHICULE   (vehicule_1, ponder_f_vehi, laisser_unif,
           dist_avant, dist_apres, enleve_avant, enleve_apres,
           nb_essieux) 5*(essieu_3) (essieu_2, essieu_2, essieu_1)
           (x_essieu1, x_essieu2, x_essieu3, x_essieu4,
            x_essieu5, x_essieu6, x_essieu7, x_essieu8);
```

Related commands

ESSIEU_VEHICULE ; CONVOI_VEHICULES

10.9 - CONVOI_VEHICULES

CONVOI_VEHICULES no_convoi type_convoi (pr_inutilise)

nb_vehicules $\langle \text{no\_vehicule} \rangle_{\text{nb\_vehicules}}$;

Parameters

- no_convoi: identification number assigned to the convoy.
- type_convoi: type of convoy. Can take one of the following values:
  - (convoi_inde_1): indicates that the maximum effect of the convoy is to be identified by putting the first vehicle in a position that produces the most unfavorable effect, then the second vehicle in the same way, in the rest of the study zone, and so on, for all the vehicles.
  - (convoi_inde_2): indicates that the maximum effect of the convoy is identified by studying all the combinations of the positions of the first two vehicles, and by choosing the position that produces the least favorable effect, then placing the other vehicles, one by one, in positions that produce the least favorable effect in the rest of the study zone.
- nb_vehicules: the number of vehicles in the convoy. Must be positive.
- no_vehicule: the number of a vehicle, defined on a VEHICULE command.

Functions

This command defines a convoy of independent vehicles that are introduced in a determined order. The minimum distances between the vehicles are determined when they are defined (see VEHICULE command, dist_avant and dist_apres parameters, and Figures 10.4 and 10.5). Two types of determination of the envelope effects are proposed.

Conditions of use

- A traffic may contain any number of this type of commands.

Methodological advice

- For the type_convoi variable, the chosen value (convoi_inde_1) reduces the search time, but does not match the study of all the possible combinations of vehicle positions. The choice of the value (convoi_inde_2) is less favorable, and results in longer search times.

Examples

```
$ can apply to the predefined load BCN/BCP of the ENV module (two trucks)
VEHICULE   (vehicule_1, .......);
<nb_vehicules = 2>
CONVOI_VEHICULES  (convoi_1, convoi_inde_2, pr_inutilise,
                  nb_vehicules) (nb_vehicules)*(vehicule_1);
```

Related commands

VEHICULE ; CONCENTREE

10.10 - CONCENTRATED

CONCENTREE   no_concentree   type_concentree   no_convoi
    ponder_d_conc   ponder_f_conc   coef_rech ;

Parameters

- no_concentree: identification number assigned to the concentrated load.
- type_concentree: type of transverse processing of the concentrated load. Can take one of the following values:
  - (vehicule_droi): indicates that the vehicles in the convoy are transversely fixed and that the right-hand edge of their footprint zone remains aligned with the right-hand edge of the lane in which they are traveling. The right-hand edge corresponds to the smallest ordinate, in the direction of the increasing abscissa.
  - (vehicule_cent): indicates that the vehicles in the convoy are transversely fixed in the axis of the lane in which they are traveling.
  - (vehicule_inte): indicates that the vehicles in the convoy are transversely mobile and that their footprint zone cannot exceed the limits of the lane in which they are traveling.
  - (vehicule_born): indicates that the vehicles in the convoy are transversely mobile and that their footprint zone can exceed the limits of the lane in which they are traveling. The axes of the wheels of each vehicle that are closest to the interior of the lane must, however, remain within these limits.

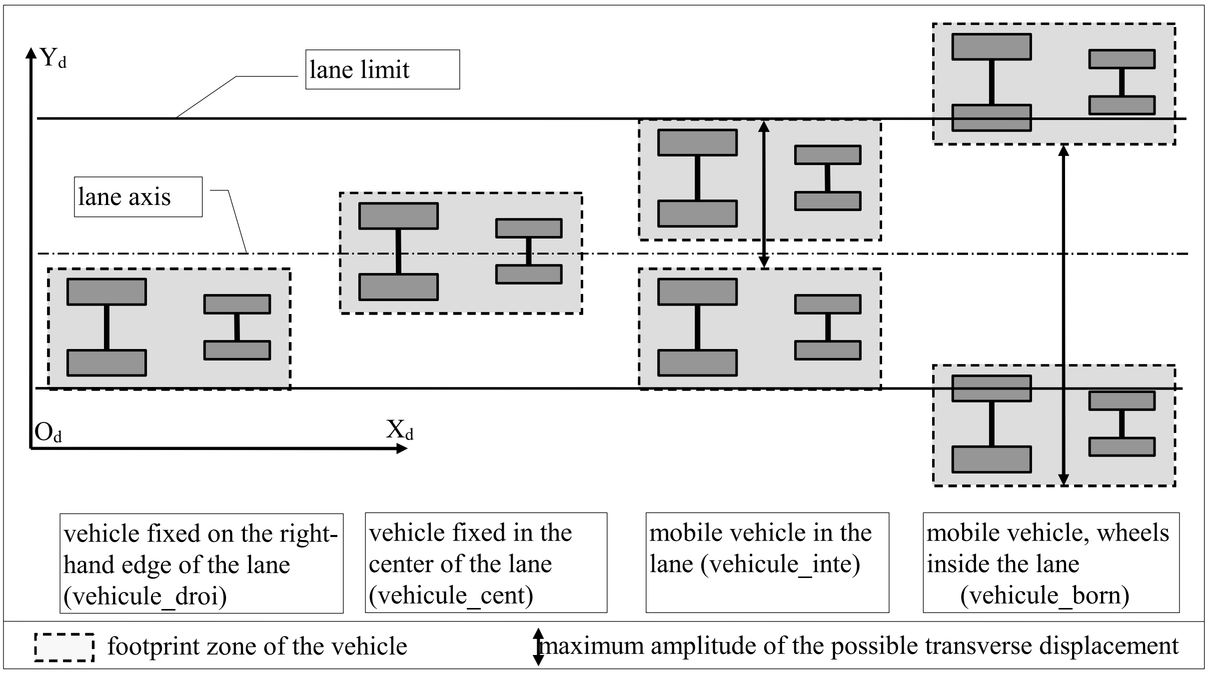


Figure 10.6 - Concentrated load. Types of transverse processing of the current vehicle

- no_convoi: number of the convoy that constitutes the concentrated load. Defined on a CONVOI_VEHICULES command.
- ponder_d_conc, ponder_f_conc: weighting coefficients to be applied to the effect of the concentrated load, whether it is respectively favorable or unfavorable. Must be positive or zero.

- coef_rech: ratio between the value of influence for the current position of a vehicle and the maximum value of the influences in the loadable zone, below which its position is not studied when looking for an envelope effect. If the value of influence at one point is lower than the product of the maximum value of influences by coef_rech, no vehicles are applied at this point. This coefficient must be positive or zero and less than or equal to 1.0.

Functions

This command defines a concentrated load made up of a convoy, to which transverse processing, lower boundaries and weighting coefficients are applied.

The type of transverse processing of the convoy specifies the positions that the vehicles can occupy in relation to the transverse limits of the lanes in which they are traveling. We verify that the transverse footprint of the vehicles does not exceed the width of the lanes.

For the values (vehicule_inte) and (vehicule_born) of the type_concentree parameter, the vehicles are displaced transversely between their limit positions, and longitudinally, along each line of influence in the lane in which they are traveling, and at the edge of the lane.

If the value of the coef_rech parameter permits, each axle of the vehicle is positioned on each point of the line of influence.

Conditions of use

- ❒ A traffic may contain any number of this type of commands.

Methodological advice

- ❒ Caution: choosing the value (vehicule_born) for the type_concentree parameter does not prevent vehicles traveling in juxtaposed lanes from overlapping.

Examples

```
$ can apply to the predefined load E_M1C1_C of the ENV module, three
$ concentrated loads defined with the same convoy, one vehicle on the
$ right edge of the road, fixed search coefficient of 0.90, favorable
$ effect weighted by 0.00, unfavorable effects weighted by decreasing
coffficients
CONVOI_VEHICULES  (convoi_1, ..., ..., ..., ...);
<ponder_d_conc = 1.00>
<ponder_f_conc = 0.00>
<coef_rech     = 0.90>
CONCENTREE   (charge_conc_1, vehicule_droi, convoi_1,
             ponder_d_conc, ponder_f_conc, coef_rech);
<ponder_d_conc = (2.00/3.00)>
CONCENTREE   (charge_conc_2, vehicule_droi, convoi_1,
             ponder_d_conc, ponder_f_conc, coef_rech);
<ponder_d_conc = (1.00/3.00)>
CONCENTREE   (charge_conc_3, vehicule_droi, convoi_1,
             ponder_d_conc, ponder_f_conc, coef_rech);
```

Related commands

CONVOI_VEHICULES ; CHARGE_VOIE

10.11 - DENSITE_UNIFORME

DENSITE_UNIFORME no_densite type_densite lg_min (pr_inutilise)
coef_b coef_c coef_d lg_imposee_max lg_imposee_min
nb_val $[\langle \text{L} \quad \text{q} \rangle_{\text{nb\_val}}]$;

Parameters

- no_densite: identification number assigned to the density of the uniform load.
- type_densite: type of variation of the density q of the uniform load (intensity by surface unit), as a function of the conventional loaded length L. Can take one of the following values:
  - (densite_tabl): density defined by a list of couples of values $\langle \text{L} \quad \text{q} \rangle_{\text{nb\_val}}$. The current density is calculated by linear interpolation in the domain covered by the list, or using the value of the nearest boundary, outside the domain of the list:

$$\begin{aligned} &q = q_i + (q_{i+1} - q_i)\left[(L - L_i)/(L_{i+1} - L_i)\right] && \text{pour :} \quad L_i < L \leq L_{i+1} \\ &q = q_1 && \text{pour :} \quad L \leq L_1 \\ &q = q_{nb\_val} && \text{pour :} \quad L > L_{nb\_val} \end{aligned} \tag{10.5}$$

  - (densite_cons) :

$$q = b \tag{10.6}$$

  - (densite_raci) :

$$q = b + c\sqrt{L} \tag{10.7}$$

  - (densite_puis) :

$$q = b / L^c \tag{10.8}$$

  - (densite_loga) :

$$q = b + c / \ln(L) \tag{10.9}$$

  - (densite_poly) :

$$q = b + c / (d + L) \tag{10.10}$$

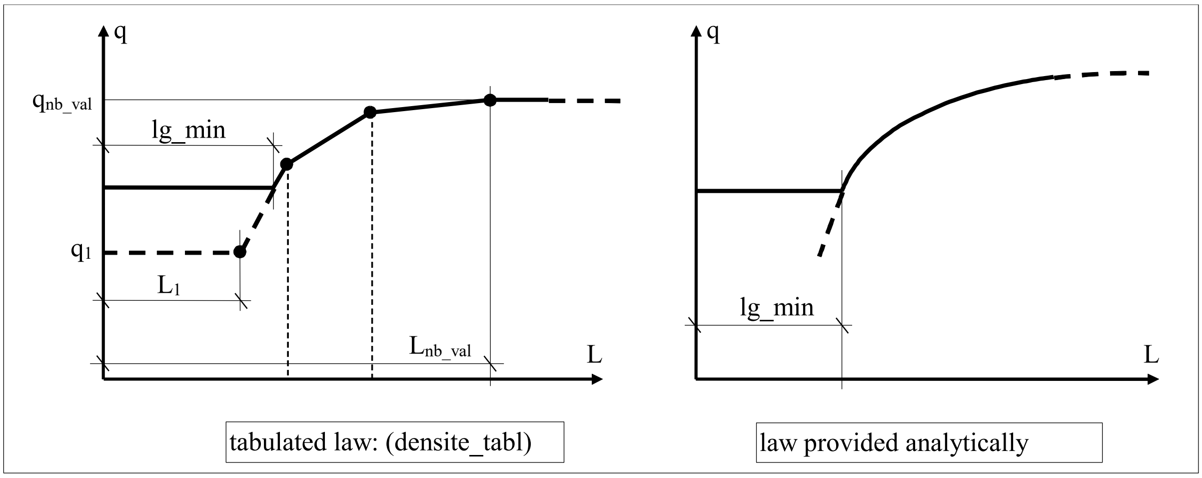


Figure 10.7 - Types of laws of variation for q(L)

- lg_min: the loaded length, below which the density of the uniform load is assumed to be constant (Figure 10.7).
- coef_b, coef_c, coef_d: coefficients b, c and d that may be used in the formulas 10.6 to 10.10.
- lg_imposee_max, lg_imposee_min: loaded length used to calculate the density q in order to study the maximum and minimum effects respectively, if the uniform load is of the imposed length type (the value of the type_longueur parameter of the corresponding UNIFORME command is (longueur_impo)). If the uniform load is of the bounded length type (the type_longueur parameter of the corresponding UNIFORME command is (longueur_born)), the density of the uniform load is bounded by the values obtained by applying the chosen density formula to these two end values. If the uniform load is of the bounded integration length type (the type_longueur parameter of the corresponding UNIFORME command is (longueur_born)), the total uniform load is bounded by the values obtained by applying the chosen density formula to these two end values.
- nb_val: the number of couples of values (L, q) to be provided. Greater than one if the density of the uniform load is of the type (densite_tabl), otherwise zero.
- L, q: conventional loaded length and density of the uniform load defining a point of discretization of the tabulated variation law q(L). The values of L must be strictly increasing.

Functions

This command defines the mode of calculation of a uniform load density, as a function of a conventional loaded length, to be used in the integration calculations of the surfaces of influence. This entity can only be invoked in a definition of a uniform load (UNIFORM command).

The uniform load density can be constant, can depend on the conventional loaded length or on the parameters provided directly.

The conventional loaded length can be the result of an effective calculation, or be imposed, depending on the option chosen in the definition of the uniform load.

The unit used corresponds to the unit defined in PH3 (by default, kdaN). The traffic loads introduced by the user must be defined in the unit chosen in the phasing.

Conditions of use

☐ A traffic may contain any number of this type of commands.

Examples

```
$ can apply to the predefined load ALPL of the ENV module
$ uniform density of the type (densite_tabl), with two points of definition
$ only nb_val is used, the other parameters are fictive values
<lg_min          = 0.00>
<coef_b          = 0.00>
<coef_c          = 0.00>
<coef_d          = 0.00>
<lg_imposee_max  = 1.00>
<lg_imposee_min  = 1.00>
<nb_val          = 2>

DENSITE_UNIFORME  (densite_1, densite_tabl, lg_min, pr_inutilise,
```

```
                   coef_b, coef_c, coef_d,
                   lg_imposee_max, lg_imposee_min, nb_val)
                   0.00 0.40   2000.00 0.00;
.....................
$ uniform load of the type (longueur_calc), max imposed lg and min imposed
$ lg_parameters of the preceding DENSITE_UNIFORME command are unused
UNIFORME  (..., ..., densite_1, longueur_calc, ..., ...);
$ can apply to the predefined load E_M1C1_C of the ENV module
$ two uniform densities of the type (densite_cons), only coef_b is used
<lg_min           = 0.00>
<coef_b           = 0.90>
<coef_c           = 0.00>
<coef_d           = 0.00>
<lg_imposee_max   = 1.00>
<lg_imposee_min   = 1.00>
<nb_val           = 0>
DENSITE_UNIFORME  (densite_1, densite_cons, lg_min, pr_inutilise, coef_b,
                   coef_c, coef_d, lg_imposee_max, lg_imposee_min, nb_val);
<coef_b = 0.25>
DENSITE_UNIFORME  (densite_2, densite_cons, lg_min, pr_inutilise, coef_b,
                   coef_c, coef_d, lg_imposee_max, lg_imposee_min, nb_val);
.....................
UNIFORME (..., ..., densite_1, longueur_calc, ..., ...);
UNIFORME (..., ..., densite_2, longueur_calc, ..., ...);

$ can apply to the predefined load AL of the ENV module, uniform
$ density type (densite_poly), only coef_b, coef_c and coef_d are used
<lg_min           = 0.00>
<coef_b           = 0.23>
<coef_c           = 36.00>
<coef_d           = 12.00>
<lg_imposee_max   = 1.00>
<lg_imposee_min   = 1.00>
<nb_val           = 0>
DENSITE_UNIFORME  (densite_1, densite_poly, lg_min, pr_inutilise, coef_b,
                   coef_c, coef_d, lg_imposee_max, lg_imposee_min, nb_val);
```

Related commands

UNIFORME

10.12 - UNIFORM

UNIFORME no_uniforme type_uniforme no_densite type_longueur
ponder_d_unif ponder_f_unif ;

Parameters

- no_uniforme: identification number assigned to the uniform load.
- type_uniforme: type of uniform load, determining the processing mode of the positive and negative areas of the surfaces of influence, depending on whether the effect they produce is favorable or unfavorable. The conventional loaded length is only calculated if the (longueur_calc) mode is chosen for the type_longueur parameter. It can take one of the following values:
  - (aires_comb): indicates that all the areas with the same sign are integrated and combined with one another, and that we are looking for the combination of areas that produces the most unfavorable effect. The conventional loaded length is recalculated for each combination.
  - (aires_cumu): indicates that all the areas of the unfavorable sign are integrated and globally aggregated, without taking the combinations of elementary areas into consideration. The calculated conventional loaded length is the mean length of the aggregated areas.
  - (aires_quel): indicates that all the areas are integrated and globally aggregated, irrespective of their sign and without taking the combinations of elementary areas into consideration (algebraic integration). The calculated conventional loaded length is the aggregate length of all the areas.
- no_densite: number of the uniform load density, defined on a DENSITE_UNIFORME command;
- type_longueur: type of determination of the *conventional* loaded lengths (named as such because they can be calculated or imposed) used to calculate the density of the uniform load. Can take one of the following values:
  - (longueur_calc): indicates that the conventional loaded lengths are those located between the points where the lines of influence pass through zero.
  - (longueur_impo): indicates that the conventional loaded lengths are fixed by the parameters lg_imposee_max and lg_imposee_min of the DENSITE_UNIFORME command depending on whether we are looking for the maximum or the minimum effect of the uniform load.
  - (longueur_born): indicates that the conventional loaded lengths are bounded by the lg_imposee_max and lg_imposee_min parameters of the DENSITE_UNIFORME command. Therefore, only the density of the load is bounded.
  - (longueur_inte_born): indicates that the conventional loaded lengths are bounded by the lg_imposee_max and lg_imposee_min parameters of the DENSITE_UNIFORME command for the assessment of the density and the integration of the length. Therefore, the total value of the load is bounded.
- ponder_d_unif, ponder_f_unif: weighting coefficients to be applied to the effect of the uniform load, whether it is respectively favorable or unfavorable. Must be positive or zero.

Functions

This command defines a uniform load applicable to a lane, always in the totality of its transverse footprint. A uniform load is defined by a mode of integration of the surfaces of influence, a mode of calculation of its density and the weighting coefficients.

Conditions of use

- A traffic may contain any number of this type of commands.

Methodological advice

- The difference in the precision of the two methods of integrating the surfaces of influence depends on their form. Surfaces with discontinuities in the longitudinal and transverse directions at the same time, must be treated with greater precision by giving one of the following values to the type_longueur parameter: (aires_comb_p), (aires_cumu_p) or (aires_quel_p).

Examples

```
$ can apply to the predefined load AL of the ENV module
<nb_val = 0>
DENSITE_UNIFORME  (densite_1, ..., ..., ..., ..., ..., ...,
                   ..., ..., nb_val);
.....................
$ search for the must unfavorable combination of areas amongst the areas
$ of the same sign, conventional loaded length recalculated for each
$ combination, standard precision, favorable and unfavorable effects
$ weighted in the same manner
<ponder_d_unif = 1.00>
<ponder_f_unif = 1.00>
UNIFORME (charge_unif_1, aires_comb, densite_1, longueur_calc,
          ponder_d_unif, ponder_f_unif);

$ can apply to the predefined load TR of the ENV module, all the areas
$ of the unfavorable sign are globally integrated and accumulated
$ calculated conventional loaded length, stardard precision
$ favorable effect weighted with a coefficient of zero
<ponder_d_unif = 1.00>
<ponder_f_unif = 0.00>
UNIFORME (charge_unif_1, aires_cumu, densite_1, longueur_calc,
          ponder_d_unif, ponder_f_unif);
```

Related commands

DENSITE_UNIFORME

10.13 - CHARGE_VOIE

CHARGE_VOIE no_charge type_charge
no_concentree no_uniforme type_enleve_unif ;

Parameters

- no_charge: identification number assigned to the load applicable to a lane.
- type_charge: type of load applicable to a lane. Can take one of the following values:
  - (charge_conc): indicates that only a concentrated load is applied to the lane.
  - (charge_unif): indicates that only a uniform load is applied to the lane.
  - (charge_gene): indicates that a concentrated load and a uniform load are applied to the lane together.
- no_concentree: the number of the concentrated load that makes up the load applicable to a lane, defined on a CONCENTREE command, if type_charge is (charge_conc) or (charge_gene), otherwise select (no_inutilise).
- no_uniform: the number of the uniform load that makes up the load applicable to a lane, defined on a UNIFORME command, if type_charge is (charge_unif) or (charge_gene), otherwise select (no_inutilise).
- type_enleve_unif: type of removal of the concentrated load. Can take one of the following values:
  - (laisser_unif): indicates that the uniform load must be left in the removal zones attached to the concentrated load (Figure 10.5). This is the value to be indicated if only one of the two loads is present (in this case, type_charge is (charge_conc) or (charge_unif)).
  - (enlever_unif): indicates that the uniform load must be removed from the removal zones attached to the concentrated load (Figure 10.5). This option is only possible if the uniform load is of the "aggregate of all areas with the unfavorable sign" or "aggregate of all areas, irrespective of their sign" type (the type_uniforme parameter of the UNIFORME command is then (aires_cumu), (aires_cumu_p), (aires_quel) or (aires_quel_p)).

Functions

This command defines a load applicable to a lane, formed by a concentrated and/or uniform load.

If there are two loads, they may be independent or linked, by declaring that the uniform load must be canceled in the removal zones attached to the concentrated load. Removal zones of the concomitant uniform load of all the vehicles in the corresponding convoy, for which the type_enleve_unif parameter of the VEHICLE command is (enlever_unif). This option only applies to the VEHICULES of the present TRAFIC for which the type_enleve_unif option is (enlever_unif).

Caution: the type_enleve_unif option only applies to the CHARGES of the current TRAFIC. It does not apply to the VEHICULES of the CHARGES of the preceding TRAFICS declared as remanent, nor to the following TRAFICS. Therefore, this option is only strictly necessary in the presence of a uniform load in the current load.

Conditions of use

- The number of loads applicable to the lanes to be defined in a traffic is linked to the type of separation of the loadable zone into lanes in the VOIES command (type_voies_0) to (type_voies_6).

Examples

```
$ can apply to the predefined load E_M1C1_C of the ENV module, three
$ concentrated loads and two uniform loads form four loads
$ applicable to the pavements (three general and one uniform)
CONCENTREE   (charge_conc_1, ..., ..., ..., ..., ...);
CONCENTREE   (charge_conc_2, ..., ..., ..., ..., ...);
CONCENTREE   (charge_conc_3, ..., ..., ..., ..., ...);
................
UNIFORME  (charge_unif_1, ..., ..., ..., ..., ...);
UNIFORME  (charge_unif_2, ..., ..., ..., ..., ...);
CHARGE_VOIE  (charge_1, charge_gene,
              charge_conc_1, charge_unif_1, laisser_unif);
CHARGE_VOIE  (charge_2, charge_gene,
              charge_conc_2, charge_unif_2, laisser_unif);
CHARGE_VOIE  (charge_3, charge_gene,
              charge_conc_3, charge_unif_2, laisser_unif);
CHARGE_VOIE  (charge_4, charge_unif,
              no_inutilise,  charge_unif_2, laisser_unif);
```

Related commands

VEHICULE ; CONCENTREE ; UNIFORME ; VOIES ; TRAFIC

10.14 - LANES

$$
\text{VOIES}\quad \text{no\_voies}
$$

$$
\left\{\begin{array}{l}
(\text{type\_voies\_0},\ \text{degres\_0}) \\[2ex]
(\text{type\_voies\_1})\ \ \text{type\_degr} \\
\left\{\begin{array}{l}
\text{CIRCULATION}\quad \left\{\begin{array}{l} \text{VOIES}\ \ \text{nb\_clas}\ \ \langle v_0\rangle_{\text{nb\_clas}}\ \ [\textit{deg\_1}] \\ \text{CLASSE}\ \ v_0\ \ v\ \ [\textit{deg\_2}] \end{array}\right\} \\
v_0\ \ v\ \ [\textit{deg\_3}]
\end{array}\right\} \\[2ex]
(\text{type\_voies\_2})\ \ \text{type\_degr} \\
\left\{\begin{array}{l}
\text{CIRCULATION}\ \ \text{CLASSE}\ \ \text{nb\_lcha}\ \ \langle \text{lcha}\rangle_{\text{nb\_lcha}}\ \ [\textit{deg\_2}] \\
\text{nb\_lcha}\ \ \langle \text{lcha}\rangle_{\text{nb\_lcha}}\ \ [\textit{deg\_3}]
\end{array}\right\} \\[2ex]
(\text{type\_voies\_3})\ \ \text{type\_degr} \\
\left\{\begin{array}{l}
\text{CIRCULATION}\quad \left\{\begin{array}{l}
\text{VOIES}\ \ \text{pas}\ \ \text{nb\_v\_max}\ \ \text{lvoi\_max} \\
\text{LA\_VOIES}\ \ \text{pas}\ \ \text{nb\_voies}\ \ \text{lvoi\_max} \\
\text{NB\_VOIES}\ \ \text{pas}\ \ \text{nb\_v\_max}\ \ \langle \text{lvoi}\rangle_{\text{nb\_v\_max}} \\
\text{CLASSE}\ \ \text{pas}\ \ \text{nb\_voies}\ \ \langle \text{lvoi}\rangle_{\text{nb\_voies}}
\end{array}\right\}\ [\textit{deg\_2}] \\
\text{pas}\ \ \text{nb\_voies}\ \ \langle \text{lvoi}\rangle_{\text{nb\_voies}}\ \ [\textit{deg\_3}]
\end{array}\right\} \\[2ex]
(\text{type\_voies\_4})\ \ \text{type\_degr} \\
\left\{\begin{array}{l}
\text{CIRCULATION}\quad \left\{\begin{array}{l}
\text{LA\_VOIES}\ \ \text{pas}\ \ \text{nb\_voies}\ \ \text{lvoi\_max} \\
\text{CLASSE}\ \ \text{pas}\ \ \text{nb\_voies}\ \ \langle \text{lvoi}\rangle_{\text{nb\_voies}}
\end{array}\right\}\ [\textit{deg\_2}] \\
\text{pas}\ \ \text{nb\_voies}\ \ \langle \text{lvoi}\rangle_{\text{nb\_voies}}\ \ [\textit{deg\_3}]
\end{array}\right\} \\[2ex]
(\text{type\_voies\_5})\ \ \text{type\_degr}\ \left\{\begin{array}{l}
\text{CIRCULATION}\quad \left\{\begin{array}{l}
\text{TROTTOIRS}\ \ [\textit{deg\_3}] \\
\left\{\begin{array}{l} \text{CHAUSSEES} \\ \text{VOIES} \end{array}\right\}\ [\textit{deg\_2}]
\end{array}\right\} \\
\text{nb\_voies}\ \ \langle \text{yvoi\_inf}\ \ \text{yvoi\_sup}\rangle_{\text{nb\_voies}}\ \ [\textit{deg\_3}]
\end{array}\right\} \\
(\text{type\_voies\_6})\ \ \text{type\_degr} \\
\left\{\begin{array}{l}
\text{CIRCULATION}\quad \left\{\begin{array}{l}
\text{VOIES}\ \ \text{pas}_1\ \ \text{pas}_2\ \ \text{nb\_v\_max}\ \ \text{lvoi\_max} \\
\text{LA\_VOIES}\ \ \text{pas}_1\ \ \text{pas}_2\ \ \text{nb\_voies}\ \ \text{lvoi\_max} \\
\text{NB\_VOIES}\ \ \text{pas}_1\ \ \text{pas}_2\ \ \text{nb\_v\_max}\ \ \langle \text{lvoi}\rangle_{\text{nb\_v\_max}} \\
\text{CLASSE}\ \ \text{pas}_1\ \ \text{pas}_2\ \ \text{nb\_voies}\ \ \langle \text{lvoi}\rangle_{\text{nb\_voies}}
\end{array}\right\}\ [\textit{deg\_2}] \\
\text{pas}_1\ \ \text{pas}_2\ \ \text{nb\_voies}\ \ \langle \text{lvoi}\rangle_{\text{nb\_voies}}\ \ [\textit{deg\_3}]
\end{array}\right\} \\
(\text{type\_voies\_7}\ )\ \ \text{type\_degr} \\
\left\{\begin{array}{l}
\text{CIRCULATION}\quad \left\{\begin{array}{l}
\text{LA\_VOIES}\ \ \text{pas}_1\ \ \text{pas}_2\ \ \text{nb\_voies}\ \ \text{lvoi\_max} \\
\text{CLASSE}\ \ \text{pas}_1\ \ \text{pas}_2\ \ \text{nb\_voies}\ \ \langle \text{lvoi}\rangle_{\text{nb\_voies}}
\end{array}\right\}\ [\textit{deg\_2}] \\
\text{pas}_1\ \ \text{pas}_2\ \ \text{nb\_voies}\ \ \langle \text{lvoi}\rangle_{\text{nb\_voies}}\ \ [\textit{deg\_3}]
\end{array}\right\}
\end{array}\right\}\ ;
$$

In this wording, replace:

- *deg_1* by: $\text{nb\_degr} \quad \left\langle \left\langle \text{coef\_degr} \right\rangle_{\text{nb\_degr}} \right\rangle_{\text{nb\_clas}}$,
- *deg_2* by: $\text{nb\_clas} \quad \text{nb\_degr} \quad \left\langle \left\langle \text{coef\_degr} \right\rangle_{\text{nb\_degr}} \right\rangle_{\text{nb\_clas}}$,
- and *deg_3* by:$\text{nb\_degr} \quad \left\langle \text{coef\_degr} \right\rangle_{\text{nb\_degr}}$.

Parameters

- no_voies: the identification number assigned to the description of the lanes.
- (type_voies_0) to (type_voies_6): indicates the type of separation of the loadable zone into lanes. Their parameters (apart from the coefficients of degressivity) are described in the subsections with the corresponding name.
- type_degr: type of processing of the transverse degressivity. See "Types of degressivity" for more details.
- deg_1, deg_2, deg_3: three possible expressions of the list of the coefficients of degressivity. See "Degressivity parameters" for more details.

(type_voies_0)

Always associated with the type of degressivity: (degres_0), indicates that a single lane covers the entire loadable zone.

For this type of separation, the following rules must be obeyed:

- no non-loadable central reservations may be defined,
- no definition of lanes or coefficients of degressivity are necessary,
- a single load must be invoked in the TRAFIC command.

(type_voies_1)

Associated with v and v_0, this type indicates that the loadable zone is split into a number nb_voies of contiguous lanes equal to the entire part of the quotient of its width, by a width v. A theoretical width v_0 is then assigned to each lane.

With the CIRCULATION VOIES option, the width v is the width of the traffic lanes, and all the values of v_0 that correspond to the nb_clas possible classes of traffic are provided in order. The circulation class then indicates the rank in this list of the value of v_0 to be used.

This type of separation corresponds to certain prescriptions of Issue 61, i.e.:

nb_voies = partie_entière of (largeur_chargeable/v)

largeur_effective_voie = largeur_chargeable/nb_voies

effet_voie_effective = effect of a load on a lane with a width of: largeur_effective_voie

effet_voie = effet_voie_effective × (v_0 / largeur_effective_voie)

The lanes are separated independently on either side of the non-loadable central reservation, if there is one, and the numbers of lanes determined for each carriageway are added to form nb_voies.

A single load is applicable to each lane and is weighted by a coefficient of degressivity extracted from a list of values provided.

For this type of separation, the following rules must be obeyed:

- the coefficients of degressivity are applied by considering the total number of lanes in the loadable zone,
- a single load must be invoked in the TRAFIC command.

(type_voies_2)

Indicates that the loadable zone is split into a number nb_voies of contiguous lanes, according to the rank that its width would occupy if it were inserted in a list of loadable widths provided in order of increasing values, lcha(1 .. nb_lcha):

- if the loadable width is less than or equal to lcha(1), nb_voies is 1,
- if the loadable width is greater than lcha(i) and less than or equal to lcha(i+1), nb_voies is i+1,
- if the loadable width is greater than lcha(nb_lcha), nb_voies is nb_lcha.

The lanes are separated independently on either side of the non-loadable central reservation, if there is one, and the numbers of lanes determined for each carriageway are added to form nb_voies.

A single load is applicable to each lane and is weighted by a coefficient of degressivity extracted from a list of values provided.

For this type of separation, the following rules must be obeyed:

- the coefficients of degressivity are applied by considering the total number of lanes in the loadable zone,
- a single load must be invoked in the TRAFIC command.

(type_voies_3)

Indicates that the loadable zone is separated into a maximum number of lanes, arranged transversely in the most unfavorable manner, each of which can receive one load.

The first lane is fixed in its most unfavorable position, then the second in the same way, in the rest of the loadable zone, and so on until no more lanes can be added, or all the lanes have been positioned.

- pas: value of the transverse displacement pitch of the lanes. This must be the greatest common divisor of the widths of the lanes and the widths of the carriageway(s).
- nb_v_max: maximum number of reference lanes.
- nb_voies: number of defined lane widths.
- lvoi_max: maximum reference width of a lane.
- lvoi: width of a lane. The widths of the lanes are provided in the order in which they are put into place.

With the CIRCULATION VOIES option, the number of lanes defined is the minimum of nb_v_max and the number of circulation lanes, and the width common to all the lanes is the minimum of lvoi_max and the width of the circulation lanes.

With the CIRCULATION LA_VOIES option, the number of lanes defined is nb_voies, and the width common to all the lanes is the minimum of lvoi_max and the width of the circulation lanes.

With the CIRCULATION NB_VOIES option, the number of defined lanes is the minimum of nb_v_max and the number of circulation lanes, and the widths of the lanes are provided (nb_v_max values of the lvoi parameter).

With the CIRCULATION CLASS option, the number of defined lanes is nb_voies, and the widths of the lanes are provided (nb_voies values of the lvoi parameter).

If the traffic has a non-loadable central reservation, no lanes can cover its footprint.

A different load is applicable to each lane and is weighted by a coefficient of degressivity extracted from a list of values provided.

For this type of separation, the following rules must be obeyed:

- the coefficients of degressivity are applied by considering the total number of lanes that can be positioned and loaded;
- the total number of loads to be invoked in the TRAFIC command must equal nb_v_max or nb_voies, depending on the respective cases of use of these variables;
- the loads must be invoked in the TRAFIC command in the same order as the widths, if a list of widths is provided.

(type_voies_4)

Indicates that the separation of the loadable zone into lanes is the same as for (type_voies_3), but the number of lanes is always fixed at nb_voies and an additional load is applied to the rest of the loadable width, made up of strips separating the lanes already in place.

Refer to the comment corresponding to (type_voies_3), excluding the CIRCULATION VOIES and CIRCULATION NB_VOIES options, for the meaning of these parameters.

For this type of separation, the following rules must be obeyed:

- the coefficients of degressivity are applied considering nb_voies+1 (the rest of the loadable width is assimilated to a single lane, from this point of view);
- the total number of loads to be invoked in the TRAFIC command must equal nb_voies+1, and the last one is applicable to the rest of the loadable width;
- the nb_voies first loads of the TRAFIC command must be invoked in the same order as the widths, if a list is provided.

(type_voies_5)

Indicates that the loadable zone is separated into a certain imposed number of lanes, whose positions and widths are fixed.

A different load is applicable to each lane and is weighted by a coefficient of degressivity extracted from a list of values provided.

With the CIRCULATION TROTTOIRS option, the number of lanes is the number of pavements of the circulation (a pavement is assimilated to a lane) and the positions and widths of the lanes are those of the circulation pavements.

With the CIRCULATION CHAUSSEES option, the number of lanes is the number of carriageways of the circulation (a carriageway is assimilated to a lane) and the positions and widths of the lanes are those of the circulation carriageways.

With the CIRCULATION VOIES option, the number of lanes is that of the circulation, and the positions and widths of the lanes are those of the circulation lanes.

- nb_voies, yvoi_inf, yvoi_sup: the number of lanes, ordinates of the lower and upper limits of a lane, defined on the transverse axis O_dY_d of the surfaces of influence. All the lanes, provided in the order of increasing ordinates, must be contained in the loadable zone, without overlapping.

For this type of separation, the following rules must be obeyed:

- no non-loadable central reservations may be defined,
- the coefficients of degressivity are applied by considering the total number of lanes in the loadable zone,
- the total number of loads to be invoked in the TRAFIC command must be greater than or equal to the number of lanes nb_voies (defined implicitly or explicitly). If it is greater than the number of lanes, the nb_voies first loads are assigned to the lanes in the order of their appearance.

(type_voies_6)

Indicates that the loadable zone is separated into a maximum number of lanes, transversely arranged in the most unfavorable way, considering all the possible combinations of the numbers of lanes present and their positions. Consequently, when two lanes are loaded, the determined position of the first lane may be different from the position obtained when loading a single lane.

- pas_1: value of the transverse displacement pitch of the lanes whose effect is obtained by interpolation, from the values calculated using pas_2. This must be the greatest common divisor of the widths of the lanes and the widths of the carriageway(s), and less than pas_2.
- pas_2: value of the transverse displacement pitch of the lanes for which the effects of the applied loads are actually calculated.

See the comment corresponding to (type_voies_3) for the meaning of the nb_v_max, nb_voies, lvoi_max and lvoi parameters.

If the traffic has a non-loadable central reservation, no lanes can cover its footprint.

A different load is applicable to each lane and is weighted by a coefficient of degressivity extracted from a list of values provided.

For this type of separation, the following rules must be obeyed:

- the coefficients of degressivity are applied considering the total number of lanes that can be positioned and loaded. The degressivity is assumed to be of the *global* type, irrespective of the value assigned to the type_degr parameter.
- the total number of loads to be invoked in the TRAFIC command must be greater than or equal to nb_v_max or nb_voies, depending on the respective use of these variables. If it is greater than the effective number of lanes nb_v, the nb_v first loads are assigned to the lanes in the order of their appearance.
- the loads of the TRAFIC command must be invoked in the same order as the widths, if a list is provided.

(type_voies_7)

Indicates that the separation of the loadable zone into lanes is the same as for (type_voies_6), but the number of lanes is always fixed at nb_voies and an additional load is applied to the rest of the loadable width, made up of strips separating the lanes already in place.

Refer to the comment corresponding to (type_voies_6), excluding the CIRCULATION VOIES and CIRCULATION NB_VOIES options, for the meaning of these parameters.

For this type of separation, the following rules must be obeyed:

- the coefficients of degressivity are applied considering nb_voies+1 (the rest of the loadable width is assimilated to a single lane, from this point of view).
- the total number of loads to be invoked in the TRAFIC command must equal nb_voies+1, and the last one is applicable to the rest of the loadable width;
- the nb_voies first loads of the TRAFIC command must be invoked in the same order as the widths, if a list is provided.

Types of degressivity

In this subsection, the term "effect" or the effet_voie$_i$ variable designates the effect of a load applied to a lane and the nb_v variable designates the number of lanes effectively taken into consideration in a calculation of the total effect.

The type_degr parameter can take different values with the following meanings:

- (degres_0): no degressivity is applied to the effects, which are simply aggregated. The coefficients of degressivity are not provided. This value can apply to types of lanes other than (type_voies_0).
- (degres_voie): the effects are classified in decreasing order of importance, and each one is weighted by the corresponding coefficient. The total effect is produced by the formula:

$$effet\_total = \sum_{i=1}^{nb\_v} coef\_degr_i \cdot effet\_voie_i \quad (10.11)$$

- (degres_g_voie): the same method is applied as for (degres_voie), but we are looking for the most unfavorable number of lanes. The total effect is given by the formula:

$$effet\_total = \max_{j=1,nb\_v} \left(\sum_{i=1}^{j} coef\_degr_i \cdot effet\_voie_i \right) \quad (10.12)$$

- (degres_glob): the coefficient corresponding to the nb_vth lane is applied to all the effects. The total effect is given by the formula:

$$effet\_total = coef\_degr_{nb\_v} \sum_{i=1}^{nb\_v} effet\_voie_i \quad (10.13)$$

- (degres_g_glob): the same method is applied as for (degres_glob), but we are looking for the most unfavorable number of lanes. The total effect is given by the formula:

$$effet\_total = \max_{j=1,nb\_v} \left(coef\_degr_j \sum_{i=1}^{j} effet\_voie_i \right) \quad (10.14)$$

Degressivity parameters

- nb_clas: the maximum number of classes for which the groups of coefficients of type *deg_1* or *deg_2* are defined. Must be greater than or equal to the maximum number of classes that the traffic can have.
- nb_degr: the number of coefficients of degressivity, positive. Must be greater than or equal to the number of lanes that can be taken into consideration, when the VOIES command refers to a CIRCULATION. When the nb_degr is greater than the nb_voies, only the nb_voies first coefficients are used to make the combinations.
- coef_degr: coefficient of degressivity. When the VOIES command refers to a CIRCULATION, a group of nb_degr coefficients is defined by class.

Functions

This command defines the transverse processing of the traffic, i.e., the mode in which its loadable width is split into lanes, the geometric characteristics of the lanes, the mode in which they are placed or the mode of transverse displacement and the manner in which they are loaded.

It also provides the mode in which the transverse degressivity and the corresponding weighting coefficients of the effects of the loads applicable to the lanes are processed.

In certain cases, it reuses the values acquired in a CIRCULATION command of the ENV module.

Conditions of use

- ☐ A traffic must only contain a single command of this type, inserted before the MODALITES command that refers to it.

Examples

```
$ width of the loadable zone covered by a single pavement
$ no pavements nor degressivity coefficients defined
VOIES  (voies_1, type_voies_0, degres_0);
................
MODALITES  (..., ..., ..., ..., ..., voies_1, ...);

$ can apply to the predefined exceptional convoy C1P/C1N of the ENV module
$ loadable zone shared transversely between two pavements, of which the
$ positions and widths are fixed, pavements merged with the CHAUSSEES
$ of the CIRCULATION command, used jointly by the ENV module
$ possible application of a different load per pavement, effects weighted
$ by given degressivity coefficients, total effect calculated as per 10.11
<nb_clas = 3>
<nb_degr = 2>
VOIES  (voies_1, type_voies_5, degres_voie)
       CIRCULATION CHAUSSEES  (nb_clas, nb_degr)
           1.00 0.00   $ n° of_degr coefficients, for class 1
           1.00 0.00   $ idem, for class 2
           1.00 0.00;  $ idem, for class 3

$ loadable zone split into no more than four pavements of the same width,
$ transversely arranged in the most favorable manner, all the
$ combinations of the pavements will be examined, pavements defined
$explicitely,
$ possible application of a different load by pavement, effects weighted
$ by given degressivity coefficients, total effect calculated as per 10.13
<pas1      = 0.50>
<pas2      = 3.00>
```

```
<nb_voies  = 4>
<lvoi      = 2.50>
<nb_degr   = 4>
VOIES   (voies_1, type_voies_6, degres_glob, pas1, pas2, nb_voies)
        (nb_voies)*(lvoi)
        (nb_degr)  1.20 1.10 0.95 0.80;
```

Related commands

LARGEUR_CHARGEABLE ; BANDE_CENTRALE ; MODALITES ; TRAFIC

10.15 - MODALITIES

MODALITES no_modalites type_modalites no_largeur no_longueur
(no_inutilise) no_voies no_bande ;

Parameters

- no_modalites: identification number assigned to the loading modalities.
- type_modalites: type of modality: non-remanent (modalites_gene) or remanent (modalites_rema).
- no_largeur: number of the loadable width of the traffic. Defined on the LARGEUR_CHARGEABLE command.
- no_longueur: number of the loadable length of the traffic. Defined on the LONGUEUR_CHARGEABLE command.
- no_voies: number of the lanes of the traffic. Defined on the VOIES command.
- no_bande: number of the central reservation of the traffic. Defined on the BANDE_CENTRALE command. Include (no_inutilise) if the traffic does not have a central reservation.

Functions

This command defines the loading modalities of a traffic, with reference to the transverse and longitudinal definition of its loadable zone, the definition of its lanes and of a possible central reservation.

If type_modalites is modalites_gene, the current TRAFIC is non-remanent, i.e., the VEHICULES of the current TRAFIC are no longer present for the following TRAFIC.

If type_modalites is modalites_rema, the VEHICULES of the current TRAFIC are remanent, which means that they remain in place and have an effect on the following TRAFIC. The vehicles in the following TRAFIC cannot occupy the removal zones of the remanent VEHICULES and any CHARGES UNIFORMES of the following TRAFIC are also removed in the longitudinal removal zone of the remanent VEHICULES. The remanence is carried over from TRAFIC to TRAFIC, up to and including the next non-remanent TRAFIC. There is no more remanence after the non-remanent TRAFIC. For example, if the successive traffics a, b and c are remanent, and the traffics d, e and f are non-remanent; the vehicles of a are remanent for b, the vehicles of a and b are remanent for c, the vehicles of a, b and c are remanent for d, and no vehicles are remanent for e and f. The order of the TRAFIC in question is the one defined in the ENVELOPPE command of PCP.

The envelope effects of the various remanent TRAFICS are successively combined with one another and with the last non-remanent TRAFIC.

Conditions of use

- A traffic must contain only one command of this type.
- The LARGEUR_CHARGEABLE, LONGUEUR_CHARGEABLE, VOIES, and possibly, BANDE_CENTRALE commands must be inserted before the MODALITES command.

Examples

```
LARGEUR_CHARGEABLE  (largeur_1)  .......;
LONGUEUR_CHARGEABLE  (longueur_1)  .......;
BANDE_CENTRALE  (bande_1)  .......;
................
VOIES  (voies_1) .......;
MODALITES  (modalites_1, modalites_gene, largeur_1, longueur_1,
            no_inutilise, voies_1, bande_1);
TRAFIC
'xxxxxxx'
        (..., modalites_1, ...)  .......;
```

Related commands

LONGUEUR_CHARGEABLE ; LARGEUR_CHARGEABLE ; BANDE_CENTRALE

VOIES ; TRAFIC

10.16 - TRAFFIC

TRAFIC titre_trafic no_trafic no_modalites nb_charges $\langle \text{no\_charge} \rangle_{\text{nb\_charges}}$;

Parameters

- titre_trafic: title given to the traffic (character string).
- no_trafic: identification number assigned to the traffic.
- no_modalites: number of the loading modalities of the traffic. Defined on the MODALITES command.
- nb_charges: number of loads applicable to the lanes that make up the traffic. Must be positive and compatible with the type of separation of the loadable width into lanes defined on the VOIES command that is called by the MODALITES command ((type_voies_0) to (type_voies_6)).
- no_charge: number of a load applicable to a lane defined on a CHARGE_VOIE command.

Functions

This command defines a traffic by a list of loads applicable to the lanes, according to certain modalities.

The coherence of the set of entities that make up the traffic is checked.

Conditions of use

- ☐ A traffic must contain only one command of this type.
- ☐ The CHARGE_VOIE and MODALITES commands must be before the TRAFIC command.

Examples

```
$ can be applied to the predefined load E_M1C1_C of the ENV module, the
$ type of split of the loadable load into pavements (4 pavements) imposes
$ the definition of four loads applicable to the pavements (number of
$pavements, plus one)
CHARGE_VOIE  (charge_1, ..., ..., ..., ...);
CHARGE_VOIE  (charge_2, ..., ..., ..., ...);
CHARGE_VOIE  (charge_3, ..., ..., ..., ...);
CHARGE_VOIE  (charge_4, ..., ..., ..., ...);
<nb_voies = 3>
VOIES  (voies_1, type_voies_4)  ..., ..., ..., ... (nb_voies)  .......;
MODALITES  (modalites_1, ..., ..., ..., ..., voies_1, ...);
<nb_charges = (nb_voies+1)>
TRAFIC
'EUROCODE 1 - CHARGE ROUTIERE - MODELE 1 - CLASSE 1 - CARACTERISTIQUE'
       (trafic_1, modalites_1, nb_charges,
       charge_1, charge_2, charge_3, charge_4);
```

Related commands

CHARGE_VOIE ; MODALITES

10.17 - END

FIN;

Functions

This command ends the definition of a traffic.

Conditions of use

- ❐ It must be at the end of each group of commands defining a traffic.

Examples

```
$ start of the file trafic.don containing two traffics
DEBUT  T_CONVU1;
................
................
TRAFIC  'CONVOI EXCEPTIONNEL TYPE U1'  .......;
FIN;

DEBUT  T_CONVU2;
................
................
TRAFIC  'CONVOI EXCEPTIONNEL TYPE U2'  .......;
FIN;
```

Related commands

DEBUT

10.18 - LIST OF PREDEFINED CONSTANTS

```
$ for unused parameters
<no_inutilise = 0>
<pr_inutilise = 0.0>

$ for the CLASSE_CIBLE command
<classe_1 = 1>
<classe_2 = 2>
<classe_3 = 3>

$ for the LONGUEUR_CHARGEABLE command
<longueur_1 = 1> $ n° of lengths

$ for the LARGEUR_CHARGEABLE command
<largeur_1 = 1> $ n° of widths

$ for the BANDE_CENTRALE command
<bande_1      = 1> $ n° of reservations
<bande_n_char = 1> $ type_of reservation
<bande_char   = 2>

$ for the ROUE_VEHICULE command
<roue_1    = 1>  $ n° of wheels
<roue_2    = 2>
<roue_3    = 3>
<roue_4    = 4>
<roue_5    = 5>
<roue_cons = 10> $ type_of wheel
<roue_loga = 11>
<roue_raci = 12>
<roue_tabl = 20>

$ for the ESSIEU_VEHICULE command
<essieu_1 = 1> $ n° of axles
<essieu_2 = 2>
<essieu_3 = 3>
<essieu_4 = 4>
<essieu_5 = 5>

$ for the VEHICULE command
<vehicule_1   = 1> $ n° of vehicles
<vehicule_2   = 2>
<vehicule_3   = 3>
<vehicule_4   = 4>
<vehicule_5   = 5>
<laisser_unif = 1> $ type_of uniform removal, same as for the command
CHARGE_VOIE
<enlever_unif = 2>

$ for the CONVOI_VEHICULES command
<convoi_1      = 1> $ n° of convoys
<convoi_2      = 2>
<convoi_3      = 3>
<convoi_4      = 4>
<convoi_5      = 5>
<convoi_inde_1 = 1> $ type_of convoy
<convoi_inde_2 = 2>

$ for the CONCENTREE command
<charge_conc_1 = 1> $ n° of concentrateds
<charge_conc_2 = 2>
<charge_conc_3 = 3>
<charge_conc_4 = 4>
<charge_conc_5 = 5>
<vehicule_born = 0> $ type_of concentrated
<vehicule_inte = 1>
<vehicule_droi = 2>
<vehicule_cent = 3>
```

```
$ for the DENSITE_UNIFORME command
<densite_1    = 1> $ n° of densities
<densite_2    = 2>
<densite_3    = 3>
<densite_4    = 4>
<densite_5    = 5>
<densite_poly = 1> $ type_of density
<densite_puis = 2>
<densite_cons = 10>
<densite_loga = 11>
<densite_raci = 12>
<densite_tabl = 20>

$ for the UNIFORME command
<charge_unif_1 = 1> $ n° of uniforms
<charge_unif_2 = 2>
<charge_unif_3 = 3>
<charge_unif_4 = 4>
<charge_unif_5 = 5>
<aires_comb    = 4> $ type_of uniform
<aires_cumu    = 5>
<aires_quel    = 6>
<longueur_calc = 0> $ type_of width
<longueur_impo = 1>

$ for the CHARGE_VOIE command
<charge_1    = 1> $ n° of loads
<charge_2    = 2>
<charge_3    = 3>
<charge_4    = 4>
<charge_5    = 5>
<charge_unif = 1> $ type_of load
<charge_conc = 2>
<charge_gene = 3>

$ for the VOIES command
<voies_1       = 1> $ n° of pavements
<type_voies_0  = 0> $ type_of pavements
<type_voies_1  = 1>
<type_voies_2  = 2>
<type_voies_3  = 3>
<type_voies_4  = 4>
<type_voies_5  = 5>
<type_voies_6  = 6>
<type_voies_7  = 7>
<degres_0      = 0> $ type_of degree
<degres_voie   = 1>
<degres_glob   = 2>
<degres_g_voie = 3>
<degres_g_glob = 4>

$ for the MODALITES command
<modalites_1    = 1> $ n° of modalities
<modalites_gene = 0> $ type_of modalities
<modalites_rema = 1> $ type_of modalities

$ for the TRAFIC command
<trafic_1 = 1> $ n° of traffics
```

Chapter 11

Dynamic study

CONTENTS

Functions of the DYN module

This module enables DYNAMIQUE analysis of the structures by MODALE superimposition or by TEMPORELLE solution. It is particularly well suited to dynamic studies of the effect of a VENT TURBULENT or a SEISME.

Dynamic analyses can be applied to any STRUCTURE activation method saved in the PH3 module during the construction.

The MASSES can be:

- calculated directly from the mechanical characteristics of the elements and the density of their component materials (initial masses);
- completely or partially defined by distribution on certain elements, or by concentration in certain nodes.

Annex A provides the full wording of the document referenced in this chapter under the condensed name: "AFPS 92" and "Eurocode 8".

Modal analysis

The DYN module is used to obtain the FREQUENCES or the PERIODES of the modes of vibration within defined intervals. If the structure is saved in a non-linear calculation, the tangential rigidity matrix is used.

It is also possible to impose the number of eigenvalues and eigenvectors to be calculated, starting with the lowest eigenvalue.

The eigenvectors (displacements) and the associated loads (support reactions, forces and stress) are MODES type effects that are considered like the other types of effects.

They can be produced or combined by the ETU module, with one another or with compatible results that are produced by the PH3 or ENV modules, or provided. They can also be viewed by the RES module.

Temporal analysis

The DYN module can be used to study the temporal behavior of a structure subjected to a set of dynamic loads. It calculates the states of the structure at the different instants of the temporal discretization. These states are defined by the displacements, speeds, accelerations, reactions, forces and stress. It also calculates the extreme states of the structure throughout the duration of the dynamic loading. The calculation can be made by linear or non-linear analysis, provided that the static state was analyzed by PH3 using the same mode.

All the results can be produced or combined by the ETU module, with one another or with compatible results that are produced by the PH3 or ENV modules, or provided. They can also be viewed by the RES module.

Spectral calculation of turbulent wind

The following must be introduced in order to study a structure subjected to turbulent wind:

- the coefficients of AMORTISSEMENT STRUCTURAL associated with the various modes,
- the AERODYNAMIQUES characteristics of the elements,
- the PROBABILISTES parameters governing the turbulent wind that is assumed to be QUASI-STATIONNAIRE (constant over time and in space).

The AMORTISSEMENT AERODYNAMIQUE coefficients associated with the different modes may be provided or calculated.

The calculation method is based on A.-G. Davenport's spectral theory of turbulent wind.

The experimentally established laws produced the interspectral densities of the wind speeds and their moments of intercorrelation, as a function of its probabilistic parameters.

The probabilistic characteristics of the forces applied to each point of the structure are determined on the basis of the aerodynamic characteristics of the elements.

The modal breakdown of the structure is used to calculate the variances and covariances of the generalized coordinates of the problem in eigenvectors.

The variances and covariances of the displacements and stress are deduced by modal superimposition, and any coefficients of aerodynamic admittance weight the effects of each mode.

Since the angles of incidence of the wind on the elements is calculated automatically, it may take any direction relative to the structure, but must remain consistent with the aerodynamic characteristics of the elements.

The response of the structure subjected to the turbulent wind is calculated in the form of standard deviations and concomitant values, *not weighted* by peak factors, which are calculated systematically, but not applied.

The results are REPONSE type effects, which are considered like the other types of effects. They may undergo the same types of processing as the modes of vibration by the ETU or RES modules.

As an option, the DYN module can automatically generate the load:

- corresponding to the effect of the MOYEN wind,
- equivalent to the displacements caused by a MODE,
- of EXTREME effect for different modes, according to a given degree of node freedom.

These are CAS DE CHARGE commands and CHARGEMENT [IDENTIQUE] NOEUDS subcommands that the PH3 module can read and apply to the structure (see Chapter 8).

Temporal calculation of turbulent wind

The spectral and interspectral densities specified for the wind can be used to determine random wind speeds whose mean spectral characteristics are close to those required. It is possible to configure the creation of these speeds so they can be adapted to the problem in hand. The quality of the stochastic wind is controlled by calculating the spectral densities and by a comparison with those introduced as data.

In the temporal calculation, where appropriate, the wind can be coupled with other dynamic actions. The analysis of the structure may be linear or non-linear.

The extremal response of the structure subjected to the turbulent wind is calculated. It automatically takes peak factors into consideration. The results are REPONSE type effects, which are considered like the other types of effects. They may undergo the same types of processing as the modes of vibration by the ETU or RES modules.

Similarly, the successive states of the structure are obtained according to a frequency defined by the user.

Seismic calculation

The seismic study is conducted on the basis of the modal analysis of the structure. The factor corresponding to the frequency of each mode is read on the standardized response spectrum or any other to be taken into consideration.

The ETU module performs the modal superimposition by quadratic combination of the eigenvectors (see Chapter 12, QUADRATIQUE option of the COMBINAISON command).

Verification of the sections

The ETU module is used to perform a study at the ultimate limit state of the beam sections, on the basis of the variances and the covariances of the loads due to the probabilistic effects of the wind or the earthquakes.

Its CDS command has an option that generates the 24 loads corresponding to the summits of a polyhedron limited to the ellipsoid of equiprobability of the loads of a section (see Chapter 12). The resistance of the section is guaranteed, when it withstands these envelope loads.

General conditions

The DYN module can only be used after the PH3 module has entered at least one saved structure in the database (see Chapter 8, SAUVER command).

Data analysis mode

The DYN module end delimiter commands can be written completely freely (from the point of view of their breakdown into lines); the label-types integrated in their presentation are purely indicative.

They are analyzed in full, in the order in which they are introduced (interpretation). The DYN module may be asked to simply verify the commands, without executing them.

Editing

Commands are echoed as their interpretation proceeds; any erroneous commands are followed by error messages.

The results file always contains a recap of the introduced entities, the product of their preprocessing, the details of the forces, produced by the calculation commands and the additional entities that were produced as an option. Certain intermediate results can also be produced as an option.

Export file

Contains all the CAS DE CHARGE commands that may be generated for the PH3 module.

Sommaire

11.1 - DYNAMIC

$$\text{DYNAMIQUE [}^{*}\left\{\begin{array}{l}\text{VERIFIER}\\ \text{TITRE}\quad \text{titre\_session}\end{array}\right\}\text{];}$$

Parameters

- titre_session: title attributed to the command file that will be reproduced at the head of the first results of the DYN module, if provided (string).

Functions

This command identifies a dynamic study file and starts a "session" using the DYN module.

In VERIFICATION mode, the DYN module checks the syntax and logic commands, without performing the calculations. The number of detectable errors is unlimited.

In EXECUTION mode (VERIFIER option not used), the validated commands are executed and certain results of the calculations can be recorded.

Conditions of use

- Must be at the beginning of the command file.

Methodological advice

- Always check the commands before starting an important calculation.

Examples

```
DYNAMIQUE;
..........
FIN;
```

This DYNAMIQUE command label over two lines enables or disables the VERIFICATION mode, removing or restoring the "$" character of the first line.

```
DYNAMIQUE     $ CHECK
TITRE ’VIADUC D’’ACCES B, CALCUL AU VENT, SESSION 1’;
..............
FIN;
```

Related commands

UNITES ; FIN

11.2 - UNITS

UNITES MASSES v_kilogrammes ;

Parameters

- v_kilogrammes: values, in kilograms, of the unit used to express the masses.

Functions

This command designates the unit used to express the masses. If it is absent, *tons* are used by default (v_kilogrammes = 1,000.0).

Conditions of use

☐ If it is present, this command must immediately follow the DYNAMIQUE command.

Examples

```
DYNAMIQUE;
$ the UNITES command is absent, by default the masses are expressed in tons
MASSES  ...
..........
```

```
DYNAMIQUE;
$ the masses are expressed in kilograms
UNITES  MASSES  1.0;
MASSES  ...
..........
```

Related commands

DYNAMIQUE ; MASSES

11.3 - MASSES

MASSES titre_masses

$$\left\{ \begin{array}{l} \underline{\text{INITIALES}} \left\{ \begin{array}{l} \underline{\text{COMPLETES}} \\ \text{DIAGONALES} \end{array} \right\} \underline{\text{STRUCTURE}} \\ \text{SUPPLEMENTAIRES} \left\{ \begin{array}{l} \text{CAS} \quad \langle \text{no\_cas} \rangle_{\text{nb\_cas}} \quad [\text{STRUCTURE} \quad \text{nom\_structure}] \\ \text{POUTRES} \quad \langle \text{no\_pou} \quad \text{mlp} \quad \text{iyp} \quad \text{izp} \rangle_{\text{nb\_pou}} \\ \text{ELEMENTS} \quad \langle \text{no\_ele} \quad \text{mle} \quad \text{iye} \quad \text{ize} \rangle_{\text{nb\_ele}} \\ \text{NOEUDS} \quad \langle \text{no\_noe} \quad \text{mpn} \quad \text{ixn} \quad \text{iyn} \quad \text{izn} \rangle_{\text{nb\_noe}} \\ \text{EXTREMITES} \quad \left\langle [\left\{ \begin{array}{c} + \\ - \end{array} \right\}]\text{no\_elx} \quad \text{mpn} \quad \text{ixn} \quad \text{iyn} \quad \text{izn} \right\rangle_{\text{nb\_elx}} \end{array} \right\} \end{array} \right\}$$

;

Parameters

The numbers of beams, elements, nodes and eccentric elements receiving assigned masses (respectively nb_pou, nb_ele, nb_noe and nb_elx) are implicitly defined by the corresponding numbers of values provided.

- titre_masses: title assigned to the group of defined masses (character string),
- no_cas: number of a PH3 loading case, from which the masses are drawn,
- nom_structure: name of the possible saved structure of the loading case,
- no_pou, mlp, iyp, izp: number of a beam, linear density and linear density inertia (in the y and z axes of the element coordinates) that are assigned when the masses are distributed on a series of beams,
- no_ele, mle, iye, ize: number of an element, linear density and linear density inertia (in the y and z axes of the local reference frame) that are assigned when the masses are distributed on a series of elements,
- no_noe, no_elx: number of a node, number of an element preceded by the *minus* sign, if the origin is designated, or the *plus* sign, or no sign, if the end is designated (this value implicitly designates the non-numbered node linked to the origin or the end of this element, by a rigid eccentricity element),
- mpn, ixn, iyn, izn: point mass and mass inertia (in the OX, OY and OZ directions of the global reference frame) that are assigned when the masses are concentrated on a series of nodes.

Functions

The INITIALES masses of the structure are the result of the mechanical characteristics of the elements and of the mass density of their component materials, This command is used to assign the initial masses to the structure and to add the masses to be distributed on a series of beams or elements to the current masses, or to concentrate them on a series of nodes (POUTRES, ELEMENTS, NOEUDS or EXTREMITES options).

With the SUPPLEMENTAIRES option, the defined masses are added to the initial masses.

With the COMPLETES option (chosen by default), the matrix of the masses of an element uses a linear interpolation function of the fields of displacement and contains the coupling terms (between degrees of node freedom) due to the eccentricity of its center of torsion in relation to its center of gravity.

The linear polar mass inertia i_x is calculated from the linear mass m_l, the linear mass inertias i_y and i_z, and the coordinates y_c and z_c of the center of torsion in the main reference frame of inertia, according to the formula:

$$i_x = i_y + i_z + (y_c^2 + z_c^2)\, m_l \tag{11.1}$$

With the DIAGONALES option, the matrix of the masses of an element is reduced to its diagonal terms.

With the STRUCTURE option (selected by default), the initial masses are calculated and distributed on the elements of the structure.

The numbers of the CAS DE CHARGES can be used to take the masses corresponding to the loads of the superstructures and other introduced elements in the form of numbered CAS DE CHARGE in PH3. The corresponding masses are obtained by dividing the force of the component z by g.

Conditions of use

- In the absence of a MASSES command, the MASSES INITIALES COMPLETES STRUCTURE command is adopted and all the elements of the structure receive their initial distributed masses by default.
- This is also the case when the first MASSES command is in SUPPLEMENTAIRES mass mode. The masses it defines are then added to the initial masses of the parts of the structure in question.
- Several MASSES commands can succeed one another in the same session. They must precede the SEQUENCES and MODES commands that use the mass values that they define.
- The INITIALES option can only be introduced in the first MASSES command.
- The COMPLETES or DIAGONALES options of a MASSES INITIALES command apply to the group of MASSES SUPPLEMENTAIRES commands that immediately follow them. Therefore, it is imperative to use the COMPLETES and DIAGONALES options together for the same structure.
- When calculating sequences of eigenvalues or modes, only the active nodes and elements of the structure called by the corresponding STRUCTURE command are taken into consideration, whether they have received assigned masses or not.
- Masses are assigned to beams, elements or nodes independently of their effective use in the calculations. Therefore, it is possible to assign "reserved" masses that can be used in various activation methods.

Methodological advice

- The masses added to the initial masses generally correspond to the masses of the parts that are not taken into consideration in the rigidity of the structure. This is often the case with spacers, bypasses, superstructures, fittings, etc.
- See also: Chapter 6, MATERIAU command, methodological advice.
- Check the total mass of the structure that is produced when processing the MODES command.

Examples

```
$ -----------------------------------------------------------------------
$ the UNITES command is absent, the masses are expressed in tons
$ the first MASSES command is in SUPPLEMENTAIRES mode, the masses
$ that it defines are added to the initial masses spread on elements
$ addition of the mass of three deviators to that of the bearing structure
$ occasional masses concentrated on a series of nodes, zero inertia
$ -----------------------------------------------------------------------
MASSES  'STRUCTURE PORTEUSE + DEVIATEURS'
    SUPPLEMENTAIRES  NOEUDS
$   no_node mpn       ixn, iyn, izn
    370     2.00       3*0.00
    870     3.00       3*0.00
    1370    2.00       3*0.00;
$ -----------------------------------------------------------------------
$ addition of the mass of the superstructures to the current masses, masses
$ spread on a series of beams, offset in the z axes
$ of their element coordinates, iyp inertia not zero, izp inertia zero
$ the COMPLETES option applies by default to the masses matrix
$ -----------------------------------------------------------------------
MASSES  'STRUCTURE PORTEUSE + DEVIATEURS + SUPERSTRUCTURES'
    SUPPLEMENTAIRES  POUTRES
$                 mlp  iyp  izp
    << (no_pou)  4.00 5.00 0.00 > no_pou = 1 A 3 >;
$ -----------------------------------------------------------------------
$ addition of the mass of the equipment to the current masses, non-offset
$ masses spread on a series of elements, zero inertia
$ -----------------------------------------------------------------------
MASSES  'STRUCTURE PORTEUSE + DEVIATEURS + SUPERSTRUCTURES + EQUIPEMENTS'
    SUPPLEMENTAIRES  ELEMENTS
$                 mle  iye  ize
    << (no_ele)  2.00 0.00 0.00 > no_ele = 10 A 1720 INC 10 >;
..............
..............
$ -----------------------------------------------------------------------
$ The initial masses of the bearing structure are declared diagonal
$ -----------------------------------------------------------------------
MASSES  'STRUCTURE PORTEUSE'
    INITIALES  DIAGONALES  STRUCTURE;
..............
```

Related commands

DYNAMIQUE ; UNITES ; SEQUENCES ; MODES ; REPONSE TEMPORELLE

CAS DE CHARGE (PH3)

11.4 - STRUCTURE

STRUCTURE nom_structure ;

Parameters

- nom_structure: name of a saved structure.

Functions

This command recalls a saved structure, on which the dynamic calculations are made.

In the rest of this chapter, $[M]$ refers to its matrix of masses and $[K]$, its matrix of rigidity.

If the structure is saved in a linear calculation, then the matrix of elastic linear rigidity is used. If it is saved in a non-linear geometric and/or mechanical calculation, the matrix of tangent rigidity is used.

Conditions of use

- ☐ The called structure must already be recorded in the database, using the SAUVER command in the PH3 module (see Chapter 8).
- ☐ The saved structure must not cause any instabilities (due, for example, to the non-placement of articulations that link the parts of the structure).
- ☐ A saved structure must be called before introducing the corresponding SEQUENCES, MODES and CALCUL TEMPOREL commands, and its effect is remanent.
- ☐ This command cannot be repeated, so several STRUCTURES cannot be defined in the same file.
- ☐ A recalled structure can be replaced in the same session in order to perform dynamic calculations at different stages of the construction.

Examples

```
$ ----------------
$ study of beam part 1
$ ----------------
STRUCTURE  FLEAU_1;
SEQUENCES  FREQUENCES  0 100 1;
..................

$ ----------------
$ study of beam part 2
$ ----------------
STRUCTURE  FLEAU_2;
SEQUENCES  FREQUENCES  0 100 1;
..................
```

Related commands

DYNAMIQUE ; SEQUENCES ; MODES ; REPONSE TEMPORELLE

11.5 - SEQUENCES

SEQUENCES

$$\left\{\begin{array}{llll}\text{FREQUENCES} & \text{frequence\_min} & \text{frequence\_max} & [\text{pas\_frequence}] \\ \text{PERIODES} & \text{periode\_min} & \text{periode\_max} & [\text{pas\_periode}]\end{array}\right\} ;$$

Parameters

- frequence_min, frequence_max: the minimum and maximum frequencies demarcating the range to be studied,
- pas_frequence: the breakdown interval of the range of frequencies,
- periode_min, period_max: the minimum and maximum periods demarcating the range to be studied,
- pas_period: the breakdown interval of the range of periods.

Functions

This command defines a range of FREQUENCES or PERIODES, to be broken down into intervals, according to a regular optional interval, and determines the number of modes of vibration of the studied structure located within each of these intervals. If there is no breakdown interval, a single interval, which is the same as the range of frequencies or periods, is used in the calculation.

The number of eigenvalues between 0.0 and a positive value λ is the number of negative pivots of the matrix $[K]-\lambda[M]$. The number of eigenvalues within the given interval is obtained by the difference, by successively assigning to λ the two values of the limits of the interval.

Conditions of use

- This command must be preceded by the MASSES and STRUCTURE commands.

Methodological advice

- This command is used to quickly obtain the approximate values of the periods and frequencies.

Examples

```
MASSES  ...
..............
$ dynamic study at the start of service
STRUCTURE  STRUSERV;
$ frequency study in the range 0 .. 100 Hertz in 1 Hertz steps
SEQUENCES  FREQUENCES  0 100 1;
$ study of periods in the range 2 .. 3 seconds (pitch absent, one interval)
SEQUENCES  PERIODES  2.0 3.0;
```

Related commands

DYNAMIQUE ; MASSES ; STRUCTURE

11.6 - MODES

MODES SOUS_ESPACE titre_modes NOMBRE nb_premiers_modes

$$\left\{\begin{array}{l} * \left\{\begin{array}{l} \text{DIMENSION} \quad \text{dim\_espace} \\ \text{PRECISION} \quad \text{eps\_vp\_1} \quad \text{eps\_vp\_2} \\ \text{ITERATIONS} \quad \text{nb\_iter\_1} \quad \text{nb\_iter\_2} \\ \text{FACTEUR\_MASSE} \quad \text{facteur} \\ \text{CONTROLER} \\ \text{NORME} \quad \left\{\begin{array}{l} \underline{\text{MASSE}} \\ \text{ELASTIQUE} \\ \text{MAX} \end{array}\right\} \\ \text{ENREGISTRER} \quad \text{no\_premier\_mode [nb\_modes\_enr]} \\ \text{EDITER} \\ \text{REUTILISER} \end{array}\right\} \\ \text{CALCULES} \end{array}\right\};$$

Parameters

- titre_modes: title given to the modes (character string),
- nb_premiers_modes: the number of requested modes. Must be positive and less than or equal to the total number of degrees of freedom of the nodes to which masses have been assigned, directly or indirectly, by at least one of their adjacent elements,
- dim_espace: the number of vectors in the subspace. Its calculated default value is: min(nb_premiers_modes + 8, 2 × nb_premiers_modes). When it is provided, it must be greater than nb_premiers_modes,
- eps_vp_1, eps_vp_2: details of the calculation of the eigenvectors for the subspace and Jacobi solution method. By default, these values are fixed at: 1.0E-8 and 1.0E-12.
- nb_iter_1, nb_iter_2: maximum number of iterations for the subspace and Jacobi solution method. By default, these values are fixed at 25 and 50.
- facteur: multiplication factor of the masses used to obtain a better conditioned system. By default, this value is 1000.0.
- no_premier_mode: record number of the first mode. The modes are saved in a database, if necessary, with the numbers no_premier_mode to no_premier_mode + nb_premiers_modes – 1, which must remain positive and less than 90_000.
- nb_modes_enr: Number of modes to save in database. By default, this number is equal to the number of calculated modes.

Functions

This command starts the calculation of an imposed number of modes of vibration of the structure being studied, starting with the smallest eigenvalue or frequency. These "first modes" are numbered consecutively from one (order numbers).

The so-called "subspace" iterative method is used, which consists of determining all the eigenvectors of an initial arbitrary subspace and converging by successive iterations towards the solution of the general problem.

The control parameters of the convergence (dimension of the subspace, precisions of the calculation, maximum numbers of iterations, multiplication factor of the masses) are fixed by default and can be modified. The dim_espace parameter is important to accelerate the convergence speed. If the user find that the iterative process is progressing slowly, the value must be increased from 10% to 20% compared to the default.

If all the convergence criteria are not met for all the requested modes, an error message is sent and the execution is interrupted.

With the CONTROLER option, the number of eigenvalues determined by the subspaces method is compared with the one obtained by the counting method. If they do not match, an error message is sent and the execution is interrupted.

With the NORME option, the eigenvectors can be standardized according to one of the following methods:

- MASSE: the product $\{X_i\}^T[M]\{X_i\}$ is fixed at 1.0 for each eigenvector $\{X_i\}$. This standard is chosen by default.
- ELASTIQUE: the product $\{X_i\}^T[K]\{X_i\}$ is fixed at 1.0 for each eigenvector $\{X_i\}$.
- MAX: the maximum value of the displacement is fixed at 1.0.

With the ENREGISTRER option, the eigenvectors and the associated loads are saved in the database with a series of consecutive numbers (usually different from their order numbers), for which the starting value is provided. These are MODES type EFFETS that can be invoked by the ETU or RES modules. By default, no modes are saved. This option **is not useful** for saving modes for a later MODAL REPONSE calculation that is the subject of the REUTILISER option or for calculating MODAL REPONSES as a result of this command because the own vectors remain in memory throughout the calculation session. The nb_modes_enr setting reduces the space occupied in the database by storing only the first significant modes of the project.

With the REUTILISER option, clean vectors are backed up for later calculation via CALCULES.

With the CALCULES option, the required vibration modes are simply extracted from the backup performed with the REUTILISER option implemented in a previous calculation.

By default, the different parameters associated with each mode (participation factor, generalized mass, etc.) are produced. With the EDITER option, the eigenvectors are also produced.

For a structure produced by a non-linear calculation, the forces in the elements are calculated from their matrices of tangential rigidity.

The tangential and normal stresses are calculated on the basis of the elastic behavior of the materials, except in cases of mechanical non-linearity. In this case, the normal stresses in the corresponding elements are impacted by an infinitely high coefficient and cannot be used.

With the CALCULES option, the required modes of vibration are simply extracted from the database.

In the absence of any optional additional options, the calculation is effective.

Conditions of use

- ☐ This command must be preceded by the STRUCTURE command.

- A mode of vibration is replaced in the database when it is redefined with the ENREGISTRER option.
- The identification number of an EFFET already saved by the PH3, ENV, DYN or ETU modules (according to the terminology used by the ETU module, see Table 12.1) cannot be used to designate a saved MODE of vibration.
- The CALCULES option can only be used if a previous calculation of MODES has been made with the REUTILISER option.

Methodological advice

- As a general rule, a small number of modes of vibration is sufficient to significantly represent the behavior of a structure, and a single MODES command may be enough to calculate them.
- Only change the control parameters of the default convergence in the absence of the convergence of the solution method.
- Always use the CONTROLER option.
- Reduce the number of registered modes to the bare minimum in terms of RES viewing of these modes.

Examples

```
$ ----------------
$ study of beam part 1
$ ----------------
STRUCTURE  FLEAU_1;
...................
$ --------------------------------------------------------------------------
$ search for the first five modes of vibration with control by counting
$ the default control parameters of the convergence are kept
$ by default, the specific vectors are standardized in MASSE and not edited
$ the associated specific vectors and solicitations are saved in the
$ database in the MODES 101 to 105
$ --------------------------------------------------------------------------
MODES  SOUS_ESPACE  'FLEAU 1, 5 PREMIERS MODES DE VIBRATION'
    NOMBRE  5  CONTROLER
    ENREGISTRER  101;

$ ----------------
$ study of beam part 2
$ ----------------
STRUCTURE  FLEAU_2;
...................
$ --------------------------------------------------------------------------
$ search for the first ten modes of vibration with control by counting
$ the default control parameters of the convergence are kept
$ the specific vectors are standardized in MAX value and edited
$ the associated specific vectors and solicitations are saved in the
$ database in the MODES 201 to 210
$ --------------------------------------------------------------------------
MODES  SOUS_ESPACE  'FLEAU 2, 10 PREMIERS MODES DE VIBRATION'
    NOMBRE  10  CONTROLER  ENREGISTRER  201  EDITER  NORME  MAX;

$ ------------------------------------------------------------------------
$ the two commands below are equivalent due to the default options
$ and values that apply; the dimensions calculated by default of the
$ subspace (with number of first_modes = 10) is: min(10+8, 2x10) = 18
$ ------------------------------------------------------------------------
MODES  SOUS_ESPACE  'TITRE'  NOMBRE  10
    CONTROLER  ENREGISTRER  11  EDITER;
..............
MODES  SOUS_ESPACE  'TITRE'  NOMBRE  10
    DIMENSION   18     PRECISIONS     1.0E-8 1.0E-12
```

```
    ITERATIONS  25 50  FACTEUR_MASSE  1000.0
    CONTROLER  NORME  MASSE  ENREGISTRER  11  EDITER;

$ -----------------------------------------------------------------------
$ Mode calculation with ULTERIEURE REUTILISATION
$ -----------------------------------------------------------------------
MODES  SOUS_ESPACE  'TITRE'  NOMBRE  1000
    CONTROLER REUTILISER;
FIN ;
$ New session with calculated modes reuse

DYNAMIQUE
..............
MODES  SOUS_ESPACE  'TITRE'  NOMBRE  1000 CALCULES

----
----
REPONSE --------------- ;

FIN ;
```

Related commands

DYNAMIQUE; MASSES; STRUCTURE; AMORTISSEMENT AERODYNAMIQUE

SELECTION MODES; ADMITTANCE MODALE; REPONSE MODALE VENT

AMORTISSEMENT STRUCTURAL; REPONSE MODALE SEISME

11.7 – MODES SELECTION

SELECTION MODES titre_modes_selectionnes

$$\left\{\begin{array}{l}
\text{NOMBRE} \quad \text{nb\_modes\_selectionnes} \\
\text{LISTE} \quad \left\langle \text{no\_mode\_selectionne} \right\rangle_{\text{nb\_modes}} \\
\text{FREQUENCES} \quad \text{frequence\_min} \quad \text{frequence\_max} \\
\text{PERIODES} \quad \text{periode\_min} \quad \text{periode\_max} \\
\text{CRITERES} \quad \left\{\begin{array}{l} \text{SUFFISANTS} \\ \text{NECESSAIRES} \end{array}\right\} \quad {}^{*}\left\{\begin{array}{l}
\text{MASSE} \quad \text{X} \quad \text{pmx} \\
\text{MASSE} \quad \text{Y} \quad \text{pmy} \\
\text{MASSE} \quad \text{Z} \quad \text{pmz} \\
\text{MASSE} \quad \text{XYZ} \quad \text{pmxyz} \\
\text{FREQUENCE} \quad \text{frequence\_sup} \\
\text{PERIODE} \quad \text{periode\_inf}
\end{array}\right\}
\end{array}\right\} ;$$

Parameters

This command applies to some of the first modes of vibration, calculated by the MODES command and consecutively numbered from 1 to nb_premiers_modes, in the order of increasing frequencies (order numbers).

The nb_modes parameter is implicitly defined by the number of corresponding values provided.

- titre_modes_selectionnes: title given to the selected modes (character string).
- nb_modes_selectionnes: the number of consecutive modes to be selected, starting with the first one included. Must be less than or equal to nb_premiers_modes.
- no_modes_selectionne: order number of a mode to be selected. Must be less than or equal to nb_premiers_modes. The entry order of the numbers is not imposed.
- frequence_min, frequence_max: the modes with frequencies within the range bounded by these minimum and maximum frequencies are selected.
- periode_min, periode_max: the modes with periods within the range bounded by these minimum and maximum periods are selected.
- pmx, pmy, pmz: minimum percentages of total mass that must be mobilized by *all* the selected modes, in the directions OX, OY or OZ of the global reference frame (20.0 means 20%).
- pmxyz: minimum percentage of total mass that must be mobilized by *all* the selected modes, irrespective of the considered direction of the global reference frame.
- frequence_sup: the modes with frequencies lower than this upper frequency limit are selected.
- periode_inf: the modes with periods greater that this lower period limit are selected.

Functions

This command is used to select the modes of vibration, calculated by the last MODES command, which will be used in the calculations of modal responses.

If none of the modes verify the specified criteria, an error message is sent and the execution is interrupted.

If the selection is made according to a simple criterion, a NOMBRE of modes, a LISTE of order numbers, a range of FREQUENCES or a range of PERIODES can be imposed.

If it is made according to several CRITERES, minimal percentages of the total MASSE to be mobilized (for all the modes, in one or several directions), an upper bound FREQUENCE or a lower bound PERIODE can be imposed.

The selected modes must meet at least one criterion if they are declared SUFFISANTS, or all the criteria, if they are declared NECESSAIRES.

If fp_i is the factor of participation of mode i in a directionΔ, M_i, the mass associated with mode i and M_t, the total mass, then for the eigenvector $\{X_i\}$ we have:

$$fp_i = \frac{\{X_i\}^T[M]\{\Delta\}}{\{X_i\}^T[M]\{X_i\}} \tag{11.2}$$

$$M_i = fp_i^2 \cdot (\{X_i\}^T[M]\{X_i\}) \tag{11.3}$$

$$M_t = \{\Delta\}^T[M]\{\Delta\} \tag{11.4}$$

And the percentage of mobilized mass is given by the ratio:

$$100 \cdot \frac{M_i}{M_t} \tag{11.5}$$

Conditions of use

- If this command is used, it must be preceded by a MODES command.
- If no selection is made, by default, all the calculated modes are selected.

Methodological advice

- Use this command to make sure that all the selected modes meet certain criteria.

Examples

```
$ ----------------
$ study of beam part 1
$ ----------------
STRUCTURE  FLEAU_1;
...................
MODES  SOUS_ESPACE  'FLEAU 1, 10 PREMIERS MODES DE VIBRATION'
    NOMBRE  10
    CONTROLER
    EDITER
    NORME  MAX
    ENREGISTRER  101;
$ ---------------------------------------------------
$ selection of a fixed NOMBRE of modes
$ ---------------------------------------------------
SELECTION  MODES
    'MODES 1 A 4'
    NOMBRE  4;
................
$ ---------------------------------------------------
$ selection of modes by LISTE or order numbers
$ ---------------------------------------------------
SELECTION  MODES
    'MODES 1, 3, 5 et 7'
    LISTE  1 3 5 7;
..................
$ ---------------------------------------------------
$ selection of modes in a range of FREQUENCES
$ ---------------------------------------------------
SELECTION  MODES
    'MODES DE FREQUENCE COMPRISE ENTRE 1 ET 2 Hertz'
    FREQUENCES  1 2;
...................
$ ---------------------------------------------------
$ selection of modes according to a single criterion
$ (NECESSAIRES is the same as SUFFISANTS in this case)
$ ---------------------------------------------------
SELECTION  MODES
    'MODES, DEPLACEMENT SELON OX, AU MOINS 0.20 DE MASSE TOTALE MOBILISEE'
    CRITERES  NECESSAIRES  MASSE  X  20;
...................
$ ----------------------------------------------------
$ selection of modes according to two NECESSAIRES criteria
$ ----------------------------------------------------
SELECTION  MODES
    'MODES, FREQUENCE < 100 Hertz ET AU MOINS 0.3 DE MASSE TOTALE MOBILISEE'
    CRITERES  NECESSAIRES
    FREQUENCE  100
    MASSE  XYZ  30;
...................
$ ----------------------------------------------------
$ selection of modes according to two SUFFISANTS criteria
$ ----------------------------------------------------
SELECTION  MODES
    'MODES, FREQUENCE < 100 Hertz OU AU MOINS 0.3 DE MASSE TOTALE MOBILISEE'
    CRITERES  SUFFISANTS
    FREQUENCE  100
    MASSE  XYZ  30;
```

Related commands

DYNAMIQUE; MODES; AMORTISSEMENT STRUCTURAL

AMORTISSEMENT AERODYNAMIQUE; ADMITTANCE MODALE

REPONSE MODALE VENT ; REPONSE MODALE SEISME

11.8 – MODAL STRUCTURAL DAMPING

AMORTISSEMENT STRUCTURAL MODAL titre_amortissement_stru

$$\left\{\begin{array}{l}\text{DECREMENT\_LOGARITHMIQUE}\\ \text{POURCENTAGE\_CRITIQUE}\end{array}\right\}$$

$$\left\{\begin{array}{l}\text{MOYEN} \quad \text{cas\_moyen}\\ \text{VARIABLE} \quad \langle \text{no\_mode} \quad \text{cas\_mode}\rangle_{\text{nb\_modes}}\end{array}\right\};$$

Parameters

This command applies to some of the first modes of vibration, calculated by the MODES command and consecutively numbered from 1 to nb_premiers_modes, in the order of increasing frequencies (order numbers).

The nb_modes parameter is implicitly defined by the number of corresponding values provided.

- titre_amortissement_stru: title given to the coefficients of structural damping (character string).
- cas_moyen: the mean coefficient of structural damping assigned to all the modes.
- no_mode: the order number of a mode. Must be less than or equal to nb_premiers_modes.
- cas_mode: coefficient of structural damping assigned to the mode no_mode.

Functions

This command is used to assign coefficients of structural damping to the different modes of vibration of the studied structure, with a view to making a modal or temporal calculation.

The DECREMENT_LOGARITHMIQUE and POURCENTAGE_CRITIQUE options designate the unit.

The damping can be assigned globally by a mean coefficient, or one coefficient can be assigned by mode.

Before assigning the damping mode by mode, all the coefficients are set to 0.0, which is their default value.

Conditions of use

- This command must be unique and must precede the corresponding REPONSE MODALE or REPONSE TEMPORELLE commands.
- It can precede the MODES command, because the existence of the selected modes is not verified.
- The list of modes provided can be partial, because the presence of a damping coefficient assigned to a mode is only verified when it is useful in a calculation of the modal response to the wind.
- This command is not exclusive with regard to the AMORTISSEMENT STRUCTURAL ELEMENTAIRE command.

Examples

```
$ -----------------------------------------------------------------------
$ a mean structural damping coefficient is assigned in the form of a
$ logarithmic decrement to all the calculated or future modes
$ -----------------------------------------------------------------------
AMORTISSEMENT  STRUCTURAL MODAL ’AMORTISSEMENT STRUCTURAL MOYEN’
    DECREMENT_LOGARITHMIQUE
    MOYEN  0.12;

$ -----------------------------------------------------------------------
$ a mean structural damping coefficient is assigned in the form of a
$ critical percentage to calculated modes 1 to 5 or future modes
$ -----------------------------------------------------------------------
AMORTISSEMENT  STRUCTURAL MODAL ’AMORTISSEMENT STRUCTURAL VARIABLE’
    POURCENTAGE_CRITIQUE
    VARIABLE
    1 2.0   2 3.0   3 4.0   4 5.0   5 6.0;
```

Related commands

DYNAMIQUE; MODES; SELECTION MODES; REPONSE MODALE VENT

REPONSE MODALE SEISME ; REPONSE TEMPORELLE

11.9 - ELEMENTARY STRUCTURAL DAMPING

AMORTISSEMENT STRUCTURAL ELEMENTAIRE titre_amortissement_stru

$$\begin{Bmatrix} \text{DECREMENT\_LOGARITHMIQUE} \\ \text{POURCENTAGE\_CRITIQUE} \end{Bmatrix}$$

$$^{**}\begin{Bmatrix} \text{MATERIAU} & \text{particule} & \text{am} \\ \text{APPUI} & \text{particule} & \text{am} \\ \text{ARTICULATION} & \text{particule} & \text{am} \end{Bmatrix} ;$$

Parameters

- titre_amortissement_stru: title given to the coefficients of structural damping (character string).
- particule: selection particle, similar to a name, of the materials, supports or articulations to which the value of the damping is applied. All the materials, articulations or supports with a name containing this particle are affected.
- am: coefficient of structural damping assigned to the material, support or articulation.

Functions

This command is used to assign coefficients of structural damping to materials, supports or articulations. These damping coefficients are an option used in the calculation of the modal response to wind and earthquakes. They are also used as an option in temporal dynamic calculations.

The calculation of the damping associated with the mode i is as follows:

$$\xi^i = \frac{\sum\limits_{j=1,n} \xi_j E_j^i}{\sum\limits_{j=1,n} E_j^i}$$

where E_j^i represents the energy dissipated by the element j for the mode i and ξ_j is the damping assigned to the element j.

The DECREMENT_LOGARITHMIQUE and POURCENTAGE_CRITIQUE options designate the unit.

By default, the materials, supports and articulations have no damping properties.

Conditions of use

- ☐ This command must be unique and must precede the corresponding REPONSE MODALE or REPONSE TEMPORELLE commands.
- ☐ This command is not exclusive with regard to the AMORTISSEMENT STRUCTURAL MODAL command.

Examples

```
$ ---------------------------------------------------------------------
$ an elementary structural damping coefficient is assigned in the form of
$ a logarithmic decrement to all the supports and materials
$ ---------------------------------------------------------------------
AMORTISSEMENT  STRUCTURAL ELEMENTAIRE ’AMORTISSEMENT STRUCTURAL’
   DECREMENT_LOGARITHMIQUE
   APPUIS         app  0.12
   MATERIAUX      bet  0.12;


$ ---------------------------------------------------------------------
$ an elementary structural damping coefficient is assigned in the form of
$ a critical percentage to the constituent parts of the structure
$ ---------------------------------------------------------------------
AMORTISSEMENT  STRUCTURAL ELEMENTAIRE ’AMORTISSEMENT STRUCTURAL’
   POURCENTAGE_CRITIQUE
   APPUIS         app1 2.0
   APPUIS         app2 3.0
   ARTICULATIONS  art1 2.0
   ARTICULATIONS  art2 3.0
   MATERIAU       bet  2.0
   ACIERS         aci  3.0;
```

Related commands

DYNAMIQUE; MODES; REPONSE MODALE VENT

REPONSE MODALE SEISME ; REPONSE TEMPORELLE

11.10 - DIRECT STRUCTURAL DAMPING

AMORTISSEMENT STRUCTURAL DIRECT titre_amortissement_stru

$$^{**}\left\{\begin{array}{l} \text{APPUI} \quad \text{particule} \quad \text{loi COMPOSANTE no\_comp} \quad \text{PARAMETRES} < p > \\ \text{ARTICULATION particule} \quad \text{loi COMPOSANTE no\_comp} \quad \text{PARAMETRES} < p > \end{array}\right\}$$

;

Parameters

- titre_amortissement_stru: title given to the coefficients of structural damping (character string).
- particule: selection particle, similar to a name, of the supports or articulations to which the value of the direct damping is applied. All the articulations or supports with a name containing this particle are affected.
- loi: name of the law defining the direct damping. The possible laws are: LINEAIRE, PARABOLIQUE, CUBIQUE and PUISSANCE.
- no_comp: number of the speed component according to which the damping acts. A number between 1 and 6.
- p: list of parameters defining the force F as a function of the speed and according to the specified component.
  - LINEAIRE law: $F = p_1.v.$
  - PARABOLIQUE law $F = p_1 v^2 + p_2 v$
  - CUBIQUE law $F = p_1 v^3 + p_2 v^2 + p_3 v$
  - PUISSANCE law $F = p_1 v^{p2}$

Functions

This command is used to assign structural damping stiffness to the degrees of freedom of supports or articulations. This structural damping stiffness is homogeneous with a force/speed or moment/(radians/s) ratio and can optionally be used in temporal dynamic calculations.

By default, the direct damping stiffness is zero.

Conditions of use

- This command must be unique and must precede the corresponding REPONSE TEMPORELLE command.
- This command is not exclusive with regard to the AMORTISSEMENT STRUCTURAL MODAL and AMORTISSEMENT STRUCTURAL ELEMENTAIRE commands.

Examples

```
$ ----------------------------------------------------------------------
$ a constant damping coefficient is assigned to the supports and
$ articulations
$ ----------------------------------------------------------------------
AMORTISSEMENT  STRUCTURAL DIRECT ’AMORTISSEMENT STRUCTURAL DIRECT’
    APPUIS       APPAMO  CONSTANT COMPOSANTE 1 PARAMETRE  100
    APPUIS       APPAMO  CONSTANT COMPOSANTE 2 PARAMETRE  100
    APPUIS       APPAMO  CONSTANT COMPOSANTE 3 PARAMETRE  100
    ARTICULATION ARTAMO  CONSTANT COMPOSANTE 1 PARAMETRE  100
    ARTICULATION ARTAMO  CONSTANT COMPOSANTE 2 PARAMETRE  100
    ARTICULATION ARTAMO  CONSTANT COMPOSANTE 3 PARAMETRE  100 ;


$ ----------------------------------------------------------------------
$ a linear damping coefficient is assigned to the supports and
$ articulations
$ ----------------------------------------------------------------------
AMORTISSEMENT  STRUCTURAL DIRECT ’AMORTISSEMENT STRUCTURAL DIRECT’
    APPUIS       APPAMO  LINEAIRE COMPOSANTE 1 PARAMETRE  100
    APPUIS       APPAMO  LINEAIRE COMPOSANTE 2 PARAMETRE  100
    APPUIS       APPAMO  LINEAIRE COMPOSANTE 3 PARAMETRE  100
    ARTICULATION ARTAMO  LINEAIRE COMPOSANTE 1 PARAMETRE  100
    ARTICULATION ARTAMO  LINEAIRE COMPOSANTE 2 PARAMETRE  100
    ARTICULATION ARTAMO  LINEAIRE COMPOSANTE 3 PARAMETRE  100 ;
```

Related commands

DYNAMIQUE; AMORTISSEMENT STRUCTURAL MODAL

AMORTISSEMENT STRUCTURAL ELEMENTAIRE; REPONSE TEMPORELLE

11.11 - AERODYNAMIC DAMPING

AMORTISSEMENT AERODYNAMIQUE titre_amortissement_aero

$$\left\{\begin{array}{l}\text{DECREMENT\_LOGARITHMIQUE}\\ \text{POURCENTAGE\_CRITIQUE}\end{array}\right\}$$

$$\text{MODAL}\quad\left\{\begin{array}{l}\text{MOYEN}\quad\text{caa\_moyen}\\ \text{VARIABLE}\quad\left\langle\text{no\_mode}\quad\text{caa\_mode}\right\rangle_{\text{nb\_modes}}\end{array}\right\}\;;$$

Parameters

This command applies to some of the first modes of vibration, calculated by the MODES command and consecutively numbered from 1 to nb_premiers_modes, in the order of increasing frequencies (order numbers).

The nb_modes parameter is implicitly defined by the number of corresponding values provided.

- titre_amortissement_aero: title given to the coefficients of aerodynamic damping (character string).
- caa_moyen: the mean coefficient of aerodynamic damping assigned to all the modes.
- no_mode: the order number of a mode. Must be less than or equal to nb_premiers_modes.
- caa_mode: coefficient of aerodynamic damping assigned to the mode no_mode.

Functions

This command is used to assign coefficients of aerodynamic damping to the different modes of vibration of the studied structure, with a view to obtaining the dynamic part of its modal response to the wind.

They are used when executing the REPONSE MODALE VENT commands that do not call on the AMORTISSEMENT AERODYNAMIQUE CALCULE option.

The DECREMENT_LOGARITHMIQUE and POURCENTAGE_CRITIQUE options designate the unit.

The damping can be assigned globally by a mean value, or one coefficient can be assigned by mode.

Before assigning the damping mode by mode, all the coefficients are set to 0.0, which is their default value.

Conditions of use

- This optional and unique command must precede the corresponding REPONSE MODALE VENT command.
- It can precede the MODES command, because the existence of the selected modes is not verified.
- The list of modes provided can be partial, because the presence of a damping coefficient assigned to a mode is only verified when it is useful in a calculation of the modal response to the wind.

Examples

```
$ -----------------------------------------------------------------------
$ a mean aerodynamic damping coefficient is assigned in the form of a
$ logarithmic decrement to all the calculated or future modes
$ -----------------------------------------------------------------------
AMORTISSEMENT  AERODYNAMIQUE  'AMORTISSEMENT AERODYNAMIQUE MOYEN'
    DECREMENT_LOGARITHMIQUE
    MODAL  MOYEN  0.12;

$ -----------------------------------------------------------------------
$ an aerodynamic damping coefficient is assigned in the form of a
$ critical percentage to calculated modes 1 to 5 or future modes
$ -----------------------------------------------------------------------
AMORTISSEMENT  AERODYNAMIQUE  'AMORTISSEMENT AERODYNAMIQUE VARIABLE'
    POURCENTAGE_CRITIQUE
    MODAL VARIABLE
    1 2.0   2 3.0   3 4.0   4 5.0   5 6.0;
```

Related commands

DYNAMIQUE ; MODES ; SELECTION MODES

REPONSE MODALE VENT

11.12 - MODAL ADMITTANCE

ADMITTANCE MODALE titre_admittance

$$\left\{\begin{array}{ll} \text{MOYENNE} & \text{cad\_moyen} \\ \text{VARIABLE} & \langle \text{no\_mode} \quad \text{cad\_mode} \rangle_{\text{nb\_modes}} \end{array}\right\} ;$$

Parameters

This command applies to some of the first modes of vibration, calculated by the MODES command and consecutively numbered from 1 to nb_premiers_modes, in the order of increasing frequencies (order numbers).

The nb_modes parameter is implicitly defined by the number of corresponding values provided.

- titre_admittance: title given to the coefficients of aerodynamic admittance (character string).
- cad_moyen: the mean coefficient of aerodynamic admittance assigned to all the modes.
- no_mode: the order number of a mode. Must be less than or equal to nb_premiers_modes.
- cad_mode: coefficient of aerodynamic admittance assigned to the mode no_mode.

Functions

This command is used to assign coefficients of aerodynamic admittance to the different modes of vibration of the studied structure, with a view to obtaining the dynamic part of its modal response to the wind or earthquakes.

These are the weighting coefficients of the effects of each mode that can be globally assigned by a mean value, or by one coefficient per mode.

Before assigning the admittance mode by mode, all the coefficients are set to 1.0, which is their default value.

Conditions of use

- This optional and unique command must precede the corresponding REPONSE MODALE VENT or REPONSE MODALE SEISME commands.
- It can precede the MODES command, because the existence of the selected modes is not verified.
- The list of modes can be partial.

Examples

```
$ -------------------------------------------------------------------------
$ a mean admittance coefficient is assigned to all the calculated modes and
$ future modes
$ -------------------------------------------------------------------------
ADMITTANCE  MODALE  'ADMITTANCE MOYENNE'
    MOYENNE  0.75;

$ -------------------------------------------------------------------------
$ an admittance coefficient is assigned to each of the calculated modes 1
$ to 5 or future modes
$ -------------------------------------------------------------------------
ADMITTANCE  MODALE  'ADMITTANCE VARIABLE'
    VARIABLE
    1 0.75
    2 0.66
    3 0.44
    4 0.65
    5 0.76;
```

Related commands

DYNAMIQUE ; MODES ; SELECTION MODES

REPONSE MODALE VENT ; REPONSE MODALE SEISME

11.13 - AERODYNAMIC CHARACTERISTICS

CARACTERISTIQUES AERODYNAMIQUES STRUCTURE titre_caracteristiques

$$[\left\{\begin{array}{c}\underline{\text{ABSOLUES}}\\ \text{RELATIVES}\end{array}\right\}]\ [\left\{\begin{array}{c}\underline{\text{COMPLETES}}\\ \text{A\_DERIVER}\end{array}\right\}]\ [\text{REPERE}\quad \left\{\begin{array}{c}\underline{\text{VENT}}\\ \text{PERMUTE}\end{array}\right\}]$$

$$**\left\{\begin{array}{l}\left\{\begin{array}{l}\text{SECTION}\quad \text{nom\_section}\\ \text{POUTRES}\quad \langle \text{no\_poutre}\rangle_{\text{nb\_poutres}}\\ \text{E\_POUTRES}\quad \langle \text{no\_element}\rangle_{\text{nb\_elements}}\\ \text{ELEMENTS}\quad \langle \text{no\_element}\rangle_{\text{nb\_elements}}\end{array}\right\}\quad \text{INCIDENCES}\quad \text{nb\_incidences}\\ \langle \text{i}\quad \text{B}\quad C_{tr}\quad C_{fr}\quad C_{po}\quad C_{mr}\quad [dC_{tr}\quad dC_{fr}\quad dC_{po}\quad dC_{mr}]\rangle_{\text{nb\_incidences}}\end{array}\right\}\ ;$$

Parameters

The numbers of beams and elements receiving assigned aerodynamic characteristics (respectively nb_poutres and nb_elements) are implicitly defined by the corresponding numbers of values provided.

- titre_caracteristiques: title assigned to the group of defined aerodynamic characteristics (character string),
- nom_section, no_poutre, no_element: name of a section-type (designating the current elements that have received an assignment), number of a beam or number of an element, the aerodynamic characteristics of which are defined.
- nb_incidences: number of angles of incidence of the wind used to tabulate the aerodynamic characteristics (limited to 50).
- i: the angle of incidence of the wind (in degrees, the amplitude of the domain of variation is limited to 360.0, the values in the list must be in increasing order) defined:
  - around the o_dx_d axis and relative to the o_dy_d axis of the generic reference frame of each of the first sections of the elements of the beams in question, if the aerodynamic characteristics are assigned to a series of POUTRES or to elements of beams by E_POUTRE (Figure 11.1, case 1).
  - around the ox axis and relative to the oy axis of the local reference frame of each of the elements concerned, if the aerodynamic characteristics are assigned to a SECTION-TYPE or to a series of ELEMENTS (Figure 11.1, case 2).
- B: width (positive) of an element in the direction determined by the angle i.
- C_{tr}, C_{fr}, C_{po}, C_{mr}: aerodynamic coefficient of drag (must be positive in the VENT reference frame), friction, lift and tilt around the axis of an element.
- dC_{tr}, dC_{fr}, dC_{po}, dC_{mr}: angular derivatives (or variation by unit of the angle of incidence of the wind expressed in *radians*) of the coefficients C_{tr}, C_{fr}, C_{po} and C_{mr}.

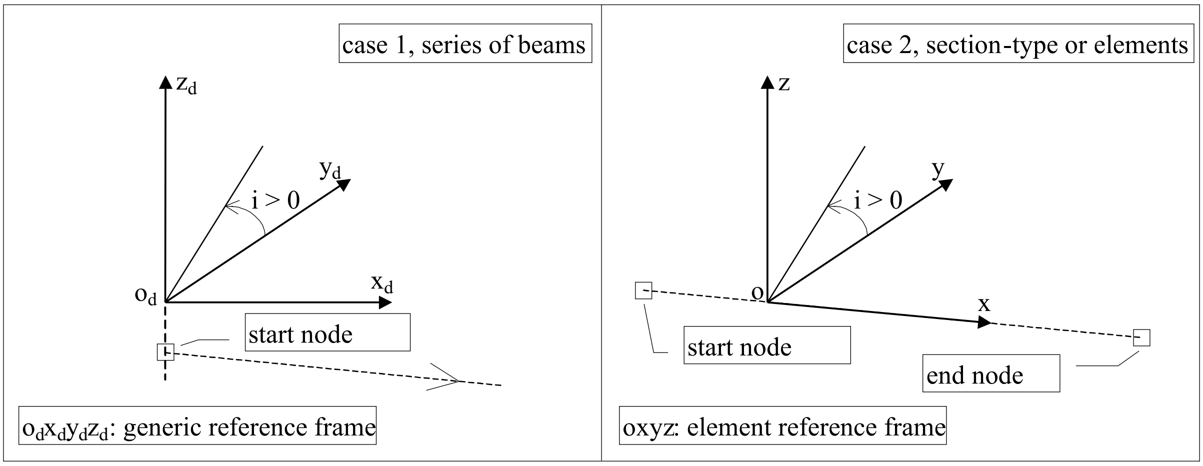


Figure 11.1 - Reference angles of incidence for aerodynamic characteristics

Functions

This command is used to define the aerodynamic characteristics of a series of elements that have been assigned to the same section-type, belonging to a list of beams or designated by their numbers.

With the ABSOLUES option (selected by default), any aerodynamic characteristics that may have been introduced previously are canceled.

With the RELATIVES option, the introduced aerodynamic characteristics can replace the former ones, and the other aerodynamic characteristics remain unchanged.

With the COMPLETES option (selected by default), the aerodynamic coefficients are provided with their angular derivatives.

With the A_DERIVER option, the aerodynamic coefficients are provided and their angular derivatives are calculated.

With the REPERE VENT option (selected by default), the aerodynamic characteristics of each element are provided in a "wind-element" type reference frame, linked to the vector representing the mean wind (known as the "mean speed vector") and to the element reference frame (Figure 11.2).

With the REPERE PERMUTE option, the aerodynamic characteristics are provided in a "permuted entity" reference frame, linked to the definition entity, according to the chosen option: SECTION, POUTRES, E_POUTRES or ELEMENTS (Figure 11.3).

The aerodynamic characteristics are tabulated for different reference angles of incidence of the wind that are defined together.

In the calculation of the effects of turbulent wind, the wind's angle of incidence on each element is determined automatically.

The associated aerodynamic characteristics are obtained by linear interpolation of the introduced values *(always in the wind reference frame, Figure 11.4)*.

When a single angle of incidence is provided, the aerodynamic characteristics of the elements are assumed to be constant, irrespective of the angle of incidence of the wind.

The forces applied by the wind to a discretization section of an element (see the VENT command) are deducted from its aerodynamic characteristics and from the mean speed vector determined in its center using the following formulas:

$$f_{tr} = \frac{1}{2}\rho \cdot U_p^2 \cdot C_{tr}(i) \cdot B(i) \tag{11.6}$$

$$f_{fr} = \frac{1}{2}\rho \cdot U_t^2 \cdot C_{fr}(i) \tag{11.7}$$

$$f_{po} = \frac{1}{2}\rho \cdot U_p^2 \cdot C_{po}(i) \cdot B(i) \tag{11.8}$$

$$f_{mr} = \frac{1}{2}\rho \cdot U_p^2 \cdot C_{mr}(i) \cdot B^2(i) \tag{11.9}$$

with:

- ρ : air density;
- U_p: norm of the projection of the mean speed vector on a plane perpendicular to the axis of the element,
- U_t: component tangential to the element of the mean speed vector,
- $f_{tr}, f_{fr}, f_{po}, f_{mr}$: drag, friction and lift force, tilting moment.

Wind reference frame

The aerodynamic characteristics of an entity (section-type, beam or element) and the corresponding aerodynamic forces can be expressed in the reference frame linked to the mean wind as follows:

- the drag component, f_{tr}, is carried by the projection of the mean speed vector on a plane perpendicular to the axis of the entity,
- the friction component, f_{fr}, is in the opposite direction of the x axis of the entity reference frame,
- the lift component, f_{po}, is the same as the trihedron (f_{tr}, f_{fr}, f_{po}), i.e., direct,
- the tilting moment, f_{mr}, around the x axis of the entity reference frame is positive, when turning from the lift axis to the drag axis.

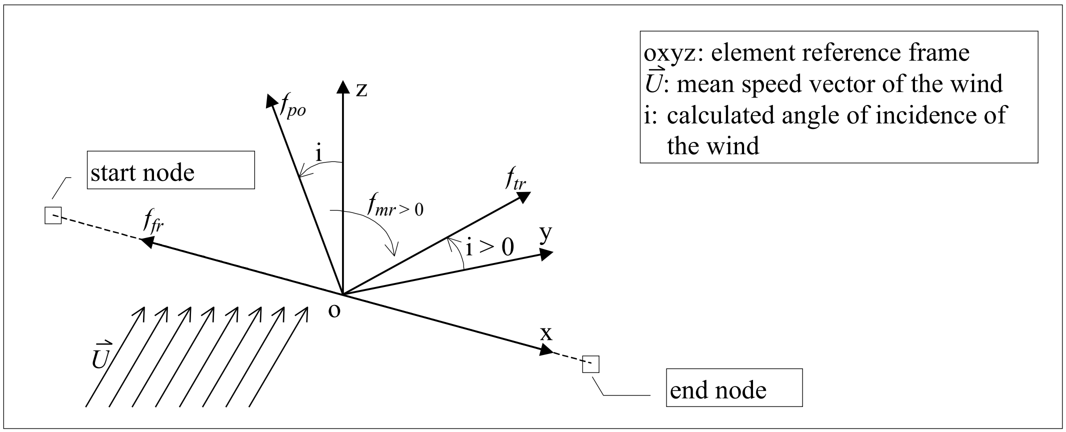


Figure 11.2 - Aerodynamic forces in the wind reference frame

The angular derivatives of the aerodynamic coefficients that may be provided are calculated in this reference frame.

Permuted reference frame

Can be expressed in a reference frame linked to the type of entity chosen for their assignment (section-type, beam or element), but permuted as follows:

- the drag component, f_{tr}, is carried by the y_e axis of the entity reference frame,
- the friction component, f_{fr}, is in the opposite direction of the x_e axis of the entity reference frame,
- the lift component, f_{po}, is the same as the trihedron (f_{tr}, f_{fr}, f_{po}), i.e., direct,
- the tilting moment, f_{mr}, around the x_e axis of the entity reference frame is positive, when turning from the lift axis to the drag axis.

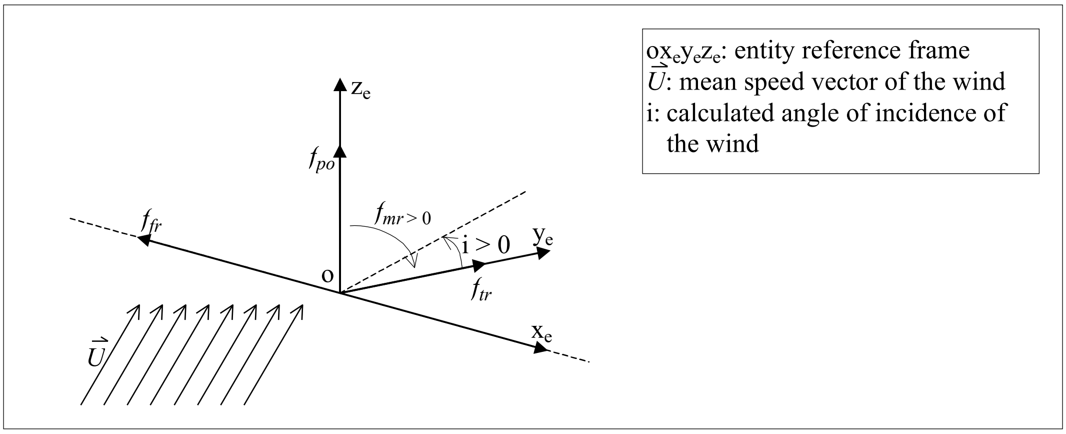


Figure 11.3 - Aerodynamic forces in the permuted reference frame

The angular derivatives of the aerodynamic coefficients that may be provided are calculated in this reference frame.

The entity reference frame (also used to the express the provided angles of incidence of the wind) are:

- the element reference frame, if aerodynamic characteristics are assigned to a section-type or to a series of elements,
- the generic reference frame (FIBRE REPERE) of the section at the start of the element, if aerodynamic characteristics are assigned to a series of beams by the POUTRES option or to beam elements by the E_POUTRE option.

The lift, drag and tilting moment of the permuted entity reference frame correspond to the homologous components of the VENTOSE program.

Link between reference frames

When the angle of incidence of the wind on the element is zero and the entity reference frame is the element reference frame, the permuted entity reference frame is the same as the wind-element reference frame.

The angular derivatives of the aerodynamic coefficients expressed in the wind reference frame and in the permuted entity reference frame are linked by the following relations:

$$\frac{\partial C_{yp}}{\partial i} = \frac{\partial C_{yv}}{\partial i} - C_{zv} \qquad (11.10)$$

$$\frac{\partial C_{zp}}{\partial i} = \frac{\partial C_{zv}}{\partial i} + C_{yv} \tag{11.11}$$

Conditions of use

- Several commands of this type can be introduced in the same session. They must precede the REPONSE MODALE VENT commands that refer to them.
- The first CARACTERISTIQUES command must use the ABSOLUES option.
- Two CARACTERISTIQUES commands using the ABSOLUES option must not immediately follow one another, since the second one would cancel out the effects of the first.
- The aerodynamic characteristics of the section-types, beams or elements can replace the previously defined values within the same definition command (possibility to "overload" values).
- The elements with aerodynamic characteristics that are all zero (apart from their width), or are not defined, are assumed not to be exposed to the wind.

Methodological advice

- The aerodynamic characteristics of non-circular sections (for which the angle of incidence of the wind and its characteristics may vary) must be tabulated with values of angles of incidence of the wind close to those that will be used in the calculations, in order to limit imprecisions due to interpolation. Information messages are shown when the aerodynamic characteristics are extrapolated.
- For circular sections, a single angle of incidence defining the aerodynamic characteristics is necessary, because they are automatically recalculated in the wind reference frame, then in the element reference frames.
- In a temporal calculation, the incidences are highly variable. To avoid extrapolation messages, it is advisable to provide values of the aerodynamic characteristics for the entire incident range, i.e., 360 degrees. If the values are not known, zero aerodynamic characteristics must be introduced at the start and the end of the corresponding incident zone.

Examples

```
$ -------------------------------------------------------------------------
$ aerodynamic characteristics defined with a single command in permuted
$ entity coordinates; three incidences of tabulation for the non-circular
$ deck described as a beam and only one incidence for the circular piles
$ described as current elements having been assigned the section-type
$ PILE; part of the pile is hidden, the zero coefficients assigned to it
$ overload the previous values
$ angles of incidence of the wind and the characteristics are expressed:
$   - in the generic coordinates for the deck (coordinate fiber)
$   - in the element coordinates for the piles
$ the ABSOLUES and COMPLETES options apply by default
$ -------------------------------------------------------------------------

CARACTERISTIQUES  AERODYNAMIQUES  STRUCTURE
    'POUR VENT QUASI-HORIZONTAL (AMPLITUDE ANGULAIRE PERMISE DE 10 DEGRES)'
    REPERE  PERMUTE
    POUTRES  1  INCIDENCES  3       $ deck
$   i     B     Ctr    Cfr    Cpo     Cmr     dCtr  dCfr  dCpo  dCmr
  -5.0  5.00  0.187  0.000  0.727  -0.106   -0.20  0.00  4.20  0.30
   0.0  5.00  0.287  0.000  0.827  -0.206   -0.10  0.00  4.50  0.40
   5.0  5.00  0.387  0.000  0.927  -0.306   -0.30  0.00  4.70  0.60
    SECTION  PILE  INCIDENCES  1     $ circular piles
$   i     B     Ctr    Cfr    Cpo     Cmr     dCtr  dCfr  dCpo  dCmr
   0.0  2.0    2.5    0.0    0.0     0.0      0.0   0.0   2.5   0.0
    ELEMENTS  1 A 10  INCIDENCES  1 $ hidden pile elements
$   i     B     Ctr    Cfr    Cpo     Cmr     dCtr  dCfr  dCpo  dCmr
   0.0  1.0    0.0    0.0    0.0     0.0     0.0   0.0   0.0   0.0;

$ -------------------------------------------------------------------------
$ same as above for a temporal calculation
$ two incidences have been added at the start and end of range for the deck
$ with force values of zero
$ -------------------------------------------------------------------------

CARACTERISTIQUES  AERODYNAMIQUES  STRUCTURE
    'POUR VENT QUASI-HORIZONTAL (AMPLITUDE ANGULAIRE PERMISE DE 10 DEGRES)'
    REPERE  PERMUTE
    POUTRES  1  INCIDENCES  7       $ deck
$   i     B     Ctr    Cfr    Cpo     Cmr     dCtr  dCfr  dCpo  dCmr
-179.9  5.00  0.000  0.000  0.000   0.000    0.00  0.00  0.00  0.00
  -6.0  5.00  0.000  0.000  0.000   0.000    0.00  0.00  0.00  0.00
  -5.0  5.00  0.187  0.000  0.727  -0.106   -0.20  0.00  4.20  0.30
   0.0  5.00  0.287  0.000  0.827  -0.206   -0.10  0.00  4.50  0.40
   5.0  5.00  0.387  0.000  0.927  -0.306   -0.30  0.00  4.70  0.60
   6.0  5.00  0.000  0.000  0.000   0.000    0.00  0.00  0.00  0.00
 179.9  5.00  0.000  0.000  0.000   0.000    0.00  0.00  0.00  0.00
    SECTION  PILE  INCIDENCES  1     $ circular piles
$   i     B     Ctr    Cfr    Cpo     Cmr     dCtr  dCfr  dCpo  dCmr
   0.0  2.0    2.5    0.0    0.0     0.0      0.0   0.0   2.5   0.0
    ELEMENTS  1 A 10  INCIDENCES  1 $ hidden pile elements
$   i     B     Ctr    Cfr    Cpo     Cmr     dCtr  dCfr  dCpo  dCmr
   0.0  1.0    0.0    0.0    0.0     0.0     0.0   0.0   0.0   0.0;
```

Related commands

DYNAMIQUE ; ACTION_STOCHASTIQUE_VENT ; REPONSE MODALE VENT

ACTION STOCHASTIQUE VENT ; REPONSE TEMPORELLE

11.14 - WIND

VENT titre_vent

$$+\left\{\begin{array}{l}
\text{MASSE\_VOLUMIQUE\_AIR} \quad \rho \\
\text{Z\_SOL} \quad z_s \\
\text{RUGOSITE\_SITE} \quad z_0 \\
\text{DISTANCE\_OBSTACLES} \quad d \\
\text{ECARTS\_TYPES} \quad \sigma_u \quad \sigma_v \quad \sigma_w \\
\text{Z\_TURBULENCE} \quad z_t \\
\text{ECHELLES\_TURBULENCE} \quad L_x^u \quad L_x^v \quad L_x^w \quad L_y^u \quad L_y^v \quad L_y^w \quad L_z^u \quad L_z^v \quad L_z^w \\
\text{VARIATION\_TURBULENCE} \quad E_x^u \quad E_x^v \quad E_x^w \quad E_y^u \quad E_y^v \quad E_y^w \quad E_z^u \quad E_z^v \quad E_z^w \\
\text{COEFFICIENTS\_COHERENCE} \quad C_x^u \quad C_x^v \quad C_x^w \quad C_y^u \quad C_y^v \quad C_y^w \quad C_z^u \quad C_z^v \quad C_z^w \\
\text{EXPOSANTS\_COHERENCE} \quad P_x^u \quad P_x^v \quad P_x^w \quad P_y^u \quad P_y^v \quad P_y^w \quad P_z^u \quad P_z^v \quad P_z^w \\
\text{VITESSE} \left\{\begin{array}{l}
\text{LOG} \quad \text{Z\_MESURE} \quad z_m \quad \text{VITESSE} \quad u_m \\
\text{PUISSANCE} \quad \text{Z\_MESURE} \quad z_m \quad \text{VITESSE} \quad u_m \quad \text{EXPOSANT} \quad p
\end{array}\right\} \\
\text{SPECTRE} \quad \text{U} \left\{\begin{array}{l}\textbf{KARMAN} \\ \textbf{KAIMAL}\end{array}\right\} \quad \text{V} \left\{\begin{array}{l}\textbf{KARMAN} \\ \textbf{KAIMAL}\end{array}\right\} \quad \text{W} \left\{\begin{array}{l}\textbf{KARMAN} \\ \textbf{KAIMAL}\end{array}\right\}
\end{array}\right\}$$

$$\left[\text{LIMITES} \quad \text{y\_min} \quad \text{y\_max}\right] \quad ;$$

Parameters

The values are provided in the OZ axis of the global reference frame. The letters u, v and w refer to the fluctuating longitudinal, horizontal and "vertical" components of the wind in its reference frame (Figure 11.4), while the formulas 11.12 to 11.21 give the meaning of certain parameters.

- titre_vent: title given to the turbulent wind (character string).
- ρ : air density,
- z_s: ground measurement,
- z_0: height (positive) measuring the roughness of the site,
- d: vertical distance to be deducted in order to take the effects of obstacles into consideration,
- $\sigma_u, \sigma_v, \sigma_w$: standard deviations of the fluctuations;
- z_t: measurement value of the turbulence,
- $L_x^u, L_x^v, L_x^w, L_y^u, L_y^v, L_y^w, L_z^u, L_z^v, L_z^w$: turbulence scales of the wind;
- $E_x^u, E_x^v, E_x^w, E_y^u, E_y^v, E_y^w, E_z^u, E_z^v, E_z^w$: exponents of the variation of the turbulence scales of the wind,
- $C_x^u, C_x^v, C_x^w, C_y^u, C_y^v, C_y^w, C_z^u, C_z^v, C_z^w$: coefficients of coherence of the wind,
- $P_x^u, P_x^v, P_x^w, P_y^u, P_y^v, P_y^w, P_z^u, P_z^v, P_z^w$: exponents of the coefficients of coherence of the wind,
- z_m, u_m: value of the point of measurement and the average speed of the wind,
- p: exponent of the variation in power of the average speed of the wind,

- y_min, y_max: minimum and maximum ordinates, in the wind reference frame, of the zone of action of the wind. All the centers of the discretization sections of elements inside this zone are taken into consideration in the calculation of the response of the structure and the nodal forces are calculated for all the nodes located within these limits. By default, the entire structure is concerned.

The L_x^u à P_z^w parameters must be positive.

Functions

This command is used to define the parameters of a turbulent wind assumed to be quasi-stationary (its probabilistic parameters are constant in time and space) to be applied to the structure in order to study its modal response.

The reference frame linked to the mean wind, known as the "wind reference frame", and the associated fluctuating components u, v and w are defined as follows:

- u is the longitudinal component of the wind, which is the same as its mean speed vector,
- v is the horizontal component of the wind,
- the "vertical" component w is the same as the trihedron (u, v, w), i.e., direct.

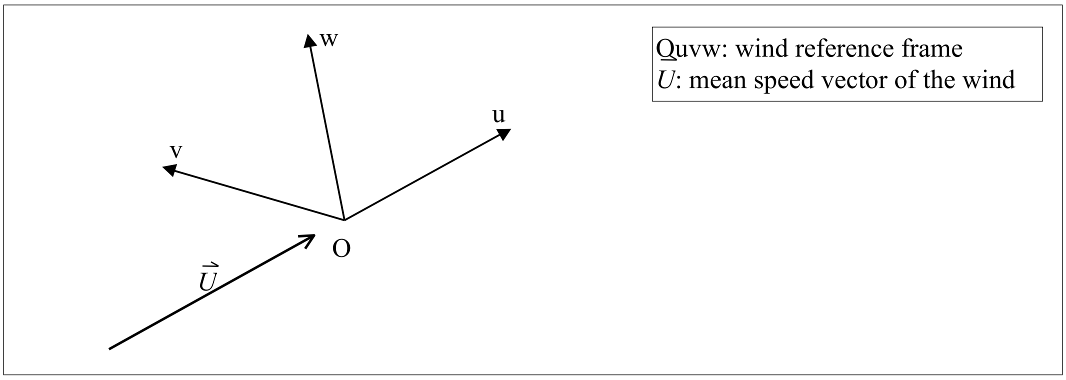


Figure 11.4 - Wind reference frame

The scales of turbulence, coefficients of coherence and certain exponents are defined for the u, v and w components of the turbulent wind as a function of the spatial variations in the homologous directions annotated with x, y and z, and which respectively correspond to the directions u, v and w.

The parameters of this command, which are, therefore, intrinsic to the wind and do not depend on the orientation of the wind reference frame in relation to the global reference frame, mainly intervene in the following relations:

- variation in LOG of the mean speed of the wind as a function of the value z:

$$\overline{U}(z) = u_m \cdot \frac{\ln\left(\dfrac{z - z_s - d}{z_0}\right)}{\ln\left(\dfrac{z_m - z_s - d}{z_0}\right)} \qquad (11.12)$$

? variation in PUISSANCE of the mean speed of the wind as a function of the value z:

$$\overline{U}(z) = u_m \cdot \left(\frac{z - z_s - d}{z_m - z_s - d} \right)^p \tag{11.13}$$

- variation in the scales of turbulence of the wind as a function of the value z:

$$L_\chi^\xi(z) = L_\chi^\xi \cdot \left(\frac{z - z_s - d}{z_m - z_s - d} \right)^{E_\chi^\xi} \tag{11.14}$$

$$avec: \quad \chi = x,y,z \;\; ; \;\; \xi = u,v,w$$

- KARMAN spectral densities as a function of the mean speed of the wind $\overline{U}$:

$$S_u(n) = \frac{4L_x^u}{\overline{U}\left[1 + 70.7\left(\frac{nL_x^u}{\overline{U}}\right)^2\right]^{5/6}} \cdot \sigma_u{}^2 \tag{11.15}$$

$$S_\xi(n) = \frac{4L_x^\xi}{\overline{U}} \frac{\left[1 + 188.4\left(\frac{2nL_x^\xi}{\overline{U}}\right)^2\right]}{\left[1 + 70.7\left(\frac{2nL_x^\xi}{\overline{U}}\right)^2\right]^{11/6}} \cdot \sigma_\xi{}^2 \tag{11.16}$$

$$avec: \quad \xi = v,w$$

- KAIMAL spectral densities as a function of the mean speed of the wind $\overline{U}$:

$$S_u(n) = \frac{17f}{n(1 + 33f)^{5/3}} \cdot \sigma_u{}^2 \tag{11.17}$$

$$S_v(n) = \frac{4.2f}{n(1 + 9.5f)^{5/3}} \cdot \sigma_v{}^2 \tag{11.18}$$

$$S_w(n) = \frac{1.4f}{n\left(1 + 5.3f^{\,5/3}\right)} \cdot \sigma_w^2 \tag{11.19}$$

$$avec: \quad f = \frac{nz}{\overline{U}}$$

- interspectral densities between two points of the structure (the exponent *v* means that the coordinates are calculated in the wind reference frame):

$$S_{\xi 1 \xi 2}(n) = \sqrt{S_{\xi 1}(n) \cdot S_{\xi 2}(n)} \cdot e^{-\frac{1}{2}\sqrt{\left[\left(\frac{x_2^v - x_1^v}{z}\right)^{P_x^\xi} C_x^\xi\right]^2 + \left[\left(\frac{y_2^v - y_1^v}{z}\right)^{P_y^\xi} C_y^\xi\right]^2 + \left[\left(\frac{z_2^v - z_1^v}{z}\right)^{P_z^\xi} C_z^\xi\right]^2} \cdot \frac{n.z}{\overline{U}}} \tag{11.20}$$

$$avec: \quad \xi = u,v,w \;\; ; \;\; z = \frac{z_1 + z_2}{2} - z_s - d$$

- intercorrelation functions of two points of the structure:

$$\rho_{\xi 1 \xi 2}(n) = e^{-\sqrt{\frac{\left(x_2^v - x_1^v\right)^2}{L_x^\xi(z_1)\,L_x^\xi(z_2)} + \frac{\left(y_2^v - y_1^v\right)^2}{L_y^\xi(z_1)\,L_y^\xi(z_2)} + \frac{\left(z_2^v - z_1^v\right)^2}{L_z^\xi(z_1)\,L_z^\xi(z_2)}}} \qquad (11.21)$$

$$avec: \quad \xi = u,v,w$$

In the relations 11.15 to 11.21, *n* is the frequency of spectral analysis of the wind.

Conditions of use

- ❒ This command must precede the REPONSE MODALE VENT command that refers to it, or the ACTION STOCHASTIQUE VENT command.
- ❒ For a REPONSE TEMPORELLE, it is possible to successively define several VENTS, provided that their limits do not join.
- ❒ Only a single VENT must be defined for a REPONSE MODALE.

Examples

```
$ -------------------------------------------------------------------------
$ definition of a quasi-stationary turbulent wind; each term of the
$ command text only appears once in a non-imposed order
$ the KARMAN and KAIMAL formulations are mixed
$ -----------------------------------------------------------------------
VENT  'VENT MESURE'
    Z_SOL                    -12.00       $ zs
    RUGOSITE_SITE            0.10         $ zo
    DISTANCE_OBSTACLES       10.00        $ d
    MASSE_VOLUMIQUE_AIR      0.00122      $ ro
    ECARTS_TYPES             9.5 9.5 7.6  $ sigmau, sigmav, sigmaw
    Z_TURBULENCE             150.00       $ zt
    ECHELLES_TURBULENCE      250  70 30   $ Lxu, Lxv, Lxw
                             130 100 80   $ Lyu, Lyv, Lyw
                              90 100 40   $ Lzu, Lzv, Lzw
    VARIATION_TURBULENCE     9*0.5        $ Exu, Exv, Exw
                                          $ Eyu, Eyv, Eyw
                                          $ Ezu, Ezv, Ezw
    COEFFICIENTS_COHERENCE   0    0 0     $ Cxu, Cxv, Cxw
                             9    9 7     $ Cyu, Cyv, Cyw
                             12  12 9     $ Czu, Czv, Czw
    EXPOSANTS_COHERENCE      9*1          $ Pxu, Pxv, Pxw
                                          $ Pyu, Pyv, Pyw
                                          $ Pzu, Pzv, Pzw
    VITESSE
       PUISSANCE
       Z_MESURE       20.00               $ zm
       VITESSE        36.00               $ um
       EXPOSANT       0.5                 $ p
    SPECTRE
    U  KARMAN
    V  KAIMAL
    W  KAIMAL;
```

Related commands

DYNAMIQUE; REPONSE MODALE VENT; ACTION STOCHASTIQUE VENT

REPONSE TEMPORELLE

11.15 - WIND MODAL RESPONSE

REPONSE MODALE VENT titre_reponse

$$\left[* \left\{ \begin{array}{l} \text{ORIENTATION\_VENT} \quad \theta_z \quad \theta_y \\ \text{LONGUEUR\_DISCRETISATION} \quad \text{lg\_max} \\ \text{SUPERPOSITION} \quad \left\{ \begin{array}{l} \underline{\text{QUADRATIQUE}} \\ \text{EXACTE} \end{array} \right\} \\ \text{AMORTISSEMENT} \quad \text{AERODYNAMIQUE} \quad \text{CALCULE} \\ \text{AMORTISSEMENT} \quad \text{STRUCTURAL} \quad \text{CALCULE} \\ \text{TOTALE} \quad [\text{EDITER}] \quad [\text{no\_totale}] \\ \text{QUASISTATIQUE} \quad [\text{EDITER}] \quad [\text{no\_quasistatique}] \\ \text{DYNAMIQUE} \quad [\text{EDITER}] \quad [\text{no\_dynamique}] \\ \text{RESULTATS} \quad \text{ELEMENTS} \\ \text{CHARGEMENT} \quad \text{MOYEN} \quad \text{no\_charge} \quad \text{FACTEUR} \quad \text{facteur} \\ \text{CHARGEMENT} \quad \left\{ \begin{array}{l} \text{MODE} \quad \text{no\_mode} \\ \text{EXTREME} \quad \left\{ \begin{array}{l} \text{NOEUD} \quad \text{no\_noeud} \\ \text{ELEMENT} \quad \text{no\_element} \end{array} \right\} \\ \qquad \text{COMPOSANTE} \quad \text{no\_comp} \end{array} \right\} \quad \text{FACTEUR} \quad \text{facteur} \\ \qquad [\text{STRUCTURE} \quad \text{nom\_structure}] \quad [\text{F\_MOYEN} \quad \text{f\_moyen}] \end{array} \right\} \right];$$

Parameters

Refer to the "Functions" section of the VENT command for the definition of the wind reference frame.

- titre_reponse: title given to the modal response (character string).
- θ_z, θ_y: the angles that orientate the wind reference frame by rotation around the OZ axis, then around the "new" OY axis, of the global reference frame. By default, these angles are zero, the Ou, Ov and Ow axes of the wind reference frame are respectively the same as the OX, OY and OZ axes of the global reference frame, and the mean speed vector is parallel to OX.

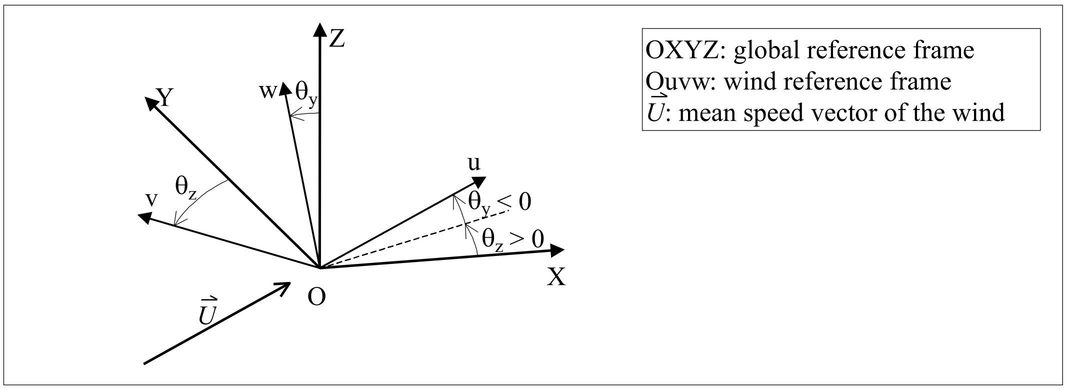


Figure 11.5 - Orientation of the wind reference frame

- lg_max: the maximum length of a discretization section of the elements when integrating the effects of the wind on the structure. By default, this value is set to 5.0.
- no_totale, no_quasistatique, no_dynamique: the possible record number of the total, quasistatic or dynamic response. Positive and less than 90_000.
- no_charge: the record number of the possible "mean wind" nodal load, to be generated for the PH3 module. Positive and less than 90_000.
- no_mode: order number of the mode used to generate the possible "mode" nodal load, which is its equivalent, for the PH3 module.
- no_noeud, no_comp: number of the node and number of the component of displacement used to generate the possible "extreme effect" nodal load for the PH3 module.
- no_element, no_comp: number of the element and number of the force component (1 to 6 for the origin and 7 to 12 for the end) used to generate the possible "extreme effect" nodal load for the PH3 module.
- facteur: weighting coefficient to be applied to the loading of the mean wind, mode or extreme effect.
- nom_structure: name of the saved structure to which the load generated for the PH3 module is applied (used on the PHASES SUITE command).
- f_moyen: weighting coefficient to be applied to the mean wind loading in cases of MODE or EXTREME loading. By default, this value is 1.

Functions

This command starts the calculation of the response of the studied structure when subjected to a quasistationary turbulent wind (predefined and of which it provides the orientation), using modal superposition and spectral analysis. This response corresponds to the standard deviation of the dynamic behavior of the structure. To have the total effect on the structure, the peak factor must be applied to the response, and the latter must be combined to the effect of the mean wind. The latter is generated on request by this command as a load case of the PCPPH3 modulus. The total effect must then be combined to the effects of the permanent loads. Both combination must be realized with PCPETU.

The effects of the wind along the elements are assessed by integration on the sections of discretization, of which the maximum length is fixed. They can be limited to an interval of ordinates.

With the SUPERPOSITION QUADRATIQUE option (selected by default), the superimposition of the effects of the selected modes is obtained by simple quadratic accumulation. The standard deviations, σ_s, of the resulting effects are obtained by the relation:

$$\sigma_s^2 = \sum C_{ii} \cdot A_i^2 \cdot s_i^2 \tag{11.22}$$

where C_{ii} is the variance of the generalized coordinates of the mode i, A_i is the coefficient of admittance of the mode i and s_i is the effect associated with the mode i (displacement, reaction, force or stress) at any point of the structure.

With the SUPERPOSITION EXACTE option, the superimposition of the effects of the selected modes calls on coefficients of correlation between the modes and effects of the wind. The standard deviation of the results are obtained by the relation:

$$\sigma_s^2 = \sum_i \sum_j C_{ij} \cdot A_i \cdot s_i \cdot A_j \cdot s_j \tag{11.23}$$

where C_{ij} is the covariance of the generalized coordinates of the modes i and j, deduced from the parameters of turbulence of the wind, using the same hypotheses as for the variance.

Methods of modal superposition used in this module are based on the section 5.2.4 of the book *Calcul dynamique des structures en zone sismique*, Alain CAPRA and Victor DAVIDOVICI, Editions Eyrolles.

With the AMORTISSEMENT AERODYNAMIQUE CALCULE option, the aerodynamic damping coefficient of each mode is calculated from the speeds of displacement that it causes. By default, the aerodynamic damping coefficients to be used are provided by the AMORTISSEMENT AERODYNAMIQUE command.

With the AMORTISSEMENT STRUCTURAL CALCULE option, the structural damping coefficient of each mode is calculated from the damping characteristics associated with the elements by the AMORTISSEMENT STRUCTURAL ELEMENTAIRE command. By default, AMORTISSEMENT STRUCTURAL MODAL is used.

The TOTALE response of the structure subjected to the turbulent wind is broken down into a QUASISTATIQUE response and a DYNAMIQUE response, which respectively express its static and dynamic behavior.

Each of these three responses can be edited (with the EDITER option) or saved in the database, if its number is provided. By default, no responses are edited or recorded. No peak factors are applied to a response that is edited or recorded.

For a structure produced by a non-linear calculation, the forces in the elements are calculated from their matrices of tangential rigidity.

The tangential and normal stresses are calculated on the basis of the elastic behavior of the materials. In cases of mechanical non-linearity, the normal stresses in the corresponding elements are impacted by an infinitely high coefficient and cannot be used.

The standard deviations of the displacements, reactions, force and stresses are always edited, and for each of the components of the corresponding load, the concomitant values of the associated components are calculated according to their relations (and edited):

$$C_s^{ab} = \sum_i \sum_j C_{ij} \cdot A_i \cdot s_i^a \cdot A_j \cdot s_j^b \quad ; \quad \sigma_s^{ab} = \frac{C_s^{ab}}{\sigma_s^{aa}} \tag{11.24}$$

where C_s^{ab} is the covariance of components a and b, s_i^a and s_j^b are the effects associated with mode i for the components a and b, σ_s^{ab} is the value of the component b concomitant with the value of component a and σ_s^{aa} is the standard deviation associated with the component a (see also 11.23 for the other ratings).

The following are also edited: for each mode, the variance of the generalized coordinates and its acceleration and the covariance of the generalized covariance, and the peak factor of each type of effect, calculated according to Davenport's method.

With the RESULTATS ELEMENTS option, the results of intermediate calculations relating to the elements are also edited. By default, they are not edited.

With the CHARGEMENT option, the nodal load equivalent to one of the following three types of effects can be generated (in the form of commands for the PH3 module that are stacked in the export file dynph3.don):

- with the MOYEN option, the effect of the mean wind, weighted by the coefficient provided, is applied to the elements in question. The nodal force vector $\left\{F_{moyen}\right\}$ is deduced from this.
- with the MODE option, the nodal load equivalent to the displacements caused by the specified mode is calculated, weighted by the coefficient provided and accumulated with the unweighted mean wind load. The nodal forces vector is expressed by:

$$\left\{F_{mode}\right\} = \text{f\_moyen}\cdot\left\{F_{moyen}\right\} + \text{facteur}\cdot\sigma_{mode}\cdot A_{mode}\cdot\left[K\right]\cdot\left\{X_{mode}\right\} \qquad (11.25)$$

- with the EXTREME option, the nodal load equivalent to the so-called "extreme" displacements vector is calculated for all the selected modes, weighted by the coefficient provided and accumulated with the unweighted mean wind load. The nodal forces vector is expressed by:

$$\left\{F_{extreme}\right\} = \text{f\_moyen}\cdot\left\{F_{moyen}\right\} + \text{facteur}\cdot\left[K\right]\cdot\left\{X_{extreme}\right\} \qquad (11.26)$$

For the CHARGEMENT EXTREME option, the displacements vector is made up of the standard deviation of the displacement of the specified node, according to the selected component, and the concomitant displacements of the other degrees of freedom. These values correspond to the total effect (quasistatic and dynamic) of the turbulent wind.

$$\left\{X_{extreme}\right\} = \left\{\begin{array}{l} \sigma_x^{1i} \\ . \\ \sigma_x^{ii} \quad \leftarrow \text{écart-type du déplacement du point d'étude} \\ . \\ \sigma_x^{ji} \quad \leftarrow \text{déplacement concomitant au déplacement du point d'étude} \\ . \\ . \\ \sigma_x^{ni} \end{array}\right\} \qquad (11.27)$$

If no STRUCTURE is specified, the current STRUCTURE is used in the PHASE SUITE command of the generated loading.

If a STRUCTURE is specified, nom_structure is used in the PHASE SUITE command of the generated loading. The current STRUCTURE is then used to again save the STRUCTURE in the generated loading. This can then be reused for a dynamic calculation. It is then possible to form a series of dynamic calculations and loadings, in which the current structure is updated according to the loading applied.

Conditions of use

- ❒ Several REPONSE MODALE commands can be introduced in the same session.
- ❒ This command must be preceded by the MODES, AMORTISSEMENT STRUCTURAL, CARACTERISTIQUES AERODYNAMIQUES and VENT commands, and possibly by the SELECTION MODES, AMORTISSEMENT AERODYNAMIQUE and ADMITTANCE MODALE commands.
- ❒ The identification number of an EFFET already saved by the PH3, ENV, DYN or ETU modules (according to the terminology used by the ETU module, see Table 12.1) cannot be used to designate a saved modal REPONSE or a MOYEN nodal wind CHARGEMENT.

- Any orientation of the wind relative to the structure is possible, but it must remain coherent with the aerodynamic characteristics of its elements. In particular, when their sections are not circular, the angles of incidence of the wind on the elements used to tabulate their aerodynamic characteristics must correspond, within the scope of the interpolation, to those that are assumed to be calculated.
- Only one wind must be defined beforehand.

Methodological advice

- The theoretical aspects of the turbulent wind are developed in the document "Vent_turbulent_theorie.pdf" provided in the subdirectory "notice" of the PCP installation directory.
- The CHARGEMENT MOYEN option applies to the structures with a linear behavior or a linearized behavior. It allows to combine the calculation of the stochastic response due to the mean wind calculated by PCP. The stochastical response is calculated according to the tangent rigidity of the structure. PCP provides the load case of mean wind in the dynph3.don file. This file must be executed by PCPPH3. The effects of the corresponding load case must then be quadratically combined by PCPETU to the stochastic response to which the peak factor is applied. The latter appears in the result file or is provided by the wind trials. The data sequence is as follow

```
$ initial phasing
facteur_pointe = 4

SAUVE structdyn
FIN

rep = 1
vm = 2

$
DYNAMIQUE;
STRUCTURE structdyn;
REPONSE  MODALE  VENT
TOTALE rep
CHARGEMENT MOYEN vm FACTEUR 1
…
FIN

$ mean wind loading of the dynph3.don file
PHASAGE SUITE structdyn
CAS DE CHARGE vm
…
FIN

PCPETU ;
…
COMBINAISON QUADRATIQUE
…
COEFFICIENT facteur_pointe REPONSE rep STRUCTURE  structdyn
COEFFICIENT 1              CHARGE  vm  STRUCTURE  structdyn
COEFFICIENT 1              ETAT     0  STRUCTURE  structdyn;
…
```

FIN

- The nodal mode or extreme force loadings are reserved for the non-linear calculations.

- The options EXTREME and MODE are reserved for the non-linear calculations. The stochastic response is calculated using the tangent rigidity and a concurrent nodal loading case is deduced which can be applied to the structure by the PH3 modulus, considering the non-linear behavior of the structure. To make correct use of the EXTREME or MODE options with a reference STRUCTURE, you must first make a double backup in PH3. One of the two backups will appear in the STRUCTURE command of the DYN module and saved STRUCTURE (SAUVE nom_structure) in the generated loading. The other will appear in the CHARGEMENT command and will be used as a reference STRUCTURE for the generated loading (PHASE SUITE). The reference STRUCTURE remains unchanged from one calculation to another, while the STRUCTURE used for the dynamic calculation is updated. The data sequence is as follows:

```
$ initial phasing
SAUVE structref
SAUVE structdyn
FIN

$ Current step
DYNAMIQUE;
STRUCTURE structdyn;
REPONSE  MODALE  VENT
…
CHARGEMENT EXTREME … STRUCTURE structref
…
FIN
$ extreme loading of the file dynph3.don
PHASAGE SUITE structref
CAS DE CHARGE
…
SAUVE structdyn
FIN
```

Examples

```
$ ------------------------------------------------------------------------
$ modal response of the structure in a quasi-stationary turbulent
$ horizontal wind, at 45.0 degrees relative to the global OX axis
$ the length of the discretization sections of elements is 5.0 by default
$ the complete structure is considered in the calculations by default
$ because the y limits of the action zone of the wind are not fixed
$ also by default, the SUPERPOSITION QUADRATIQUE option is used
$ the aerodynamic damping coefficients of the modes are calculated
$ the AMORTISSEMENT AERODYNAMIQUE command is absent; the three responses
$ (total, quasi-static and dynamic) will be saved with the numbers
$ 101 to 103 and edited, the intermediate results relating to the elements
$ are also edited; no loading is generated by the module PH3
$ ------------------------------------------------------------------------
REPONSE  MODALE  VENT  ’VENT DE NORD-EST’
$                    tetaz  tetay
   ORIENTATION_VENT  45.0   0.0
   AMORTISSEMENT  AERODYNAMIQUE  CALCULE
   TOTALE  EDITER  101
   QUASISTATIQUE EDITER  102
   DYNAMIQUE  EDITER  103
   RESULTATS  ELEMENTS;
```

```
$ ------------------------------------------------------------------------
$ modal response of the structure in a quasi-stationary turbulent
$ ascending wind at 45.0 degrees to the global OX axis and
$ 10.0 degrees around the global OY axis, itself turned by 45.0 degrees
$ the length of the discretization sections of elements is set to 2.5
$ the y limits of the action zone of the wind are fixed in wind coordinates
$ the SUPERPOSITION EXACTE option is used; the aerodynamic damping
$ coefficients of the modes are supplied by an AMORTISSEMENT
$ AERODYNAMIQUE command; only the total response is saved with
$ the number 101 and edited; no intermediate results are edited
$ the nodal loading of the mean wind, weighted by the coefficient 1.0, is
$ saved with the number 1001 and translated into commands for module PH3
$ which will use the structure STRUC_1 in association
$ ------------------------------------------------------------------------
REPONSE  MODALE  VENT  'VENT DE NORD-EST ASCENDANT'
   ORIENTATION_VENT  45.0  10.0    (tetaz, tetay)
   LONGUEUR_DISCRETISATION  2.5
   LIMITES  -2.5  15.0             (y_min, y_max)
   SUPERPOSITION  EXACTE
   TOTALE  EDITER  101
   CHARGEMENT  MOYEN  1001  FACTEUR  1.0  STRUCTURE  STRUC_1;

$ ------------------------------------------------------------------------
$ example analogous to the one above, the nodal loading of extreme effect
$ calculated for the node 1005 and the component of displacement 2 (Uy) is
$ weighted by coefficient 2.0 and translated into commands for module PH3
$ ------------------------------------------------------------------------
REPONSE  MODALE  VENT  'VENT DE NORD-EST ASCENDANT'
   ORIENTATION_VENT  45.0  10.0
   LONGUEUR_DISCRETISATION  2.5
   LIMITES  -2.5  15.0
   SUPERPOSITION  EXACTE
   TOTALE  EDITER  101
   CHARGEMENT  EXTREME  NOEUD  1005  COMPOSANTE  2  FACTEUR  2.0
   STRUCTURE  STRUC_1;
```

Related commands

DYNAMIQUE ; MODES ; SELECTION MODES

AMORTISSEMENT STRUCTURAL; AMORTISSEMENT AERODYNAMIQUE

ADMITTANCE MODALE; CARACTERISTIQUES AERODYNAMIQUES; VENT

11.16 - SPECTRUM

$$\text{SPECTRE SISMIQUE} \begin{Bmatrix} \text{HORIZONTAL} \\ \text{VERTICAL} \end{Bmatrix} \text{titre\_spectre}$$

$$\begin{Bmatrix}
\text{DEPLACEMENT} \ \langle \text{T} \ \text{u} \rangle_{\text{nb\_points}} \\
\text{ACCELERATION} \ \langle \text{T} \ \gamma \rangle_{\text{nb\_points}} \\
\text{NORMALISE} \begin{Bmatrix}
\begin{Bmatrix} \textbf{AFPS92\_E} \\ \textbf{AFPS92\_C} \end{Bmatrix}
\begin{Bmatrix}
\text{SITE} \begin{Bmatrix} \text{S0} \\ \text{S1} \\ \text{S2} \\ \text{S3} \end{Bmatrix} \\
+ \\
\text{PARAMETRES} \begin{Bmatrix} \text{TB} \ \text{tb} \\ \text{TC} \ \text{tc} \\ \text{TD} \ \text{td} \\ \text{RA} \ \text{ra} \\ \text{RM} \ \text{rm} \end{Bmatrix}
\end{Bmatrix} \\
\begin{Bmatrix} \textbf{EC8\_DF\_E} \\ \textbf{EC8\_DF\_C} \end{Bmatrix}
\begin{Bmatrix}
\text{SITE} \begin{Bmatrix} \text{SA1/SB1/SC1/SD1/SE1} \\ \text{SA2/SB2/SC2/SD2/SE2} \\ \text{FA1/FB1/FC1/FD1/FE1} \\ \text{FA2/FB2/FC2/FD2/FE2} \end{Bmatrix} \\
+ \\
\text{PARAMETRES} \begin{Bmatrix} \text{TB} \ \text{tb} \\ \text{TC} \ \text{tc} \\ \text{TD} \ \text{td} \\ \text{S} \ \text{s} \\ \text{BETA} \ \beta \end{Bmatrix}
\end{Bmatrix}
\end{Bmatrix} \text{CALAGE} \ \text{c}
\end{Bmatrix} ;$$

Parameters

The number of tabulation points (nb_points) is implicitly defined by the number of pairs of values provided.

- titre_spectre: title given to the spectrum (character string).
- T, u, γ: period, corresponding displacement or acceleration value of the spectrum, provided point by point.
- tb, tc, td: parameters common to all the standardized spectra.
- ra, rm: parameters specific to the standardized spectra of AFPS 92.
- s: ground parameter specific to the standardized spectra of Eurocode 8.
- β : lower limit parameter specific to the standardized calculation spectrum of Eurocode 8.

- c: adjustment factor of the standardized spectrum to be applied to the values that it provides, in order to obtain the maximum response values. For the standardized spectra of the AFPS, it corresponds to the nominal acceleration (which depends on the class of the bridge and the site), and for the standardized spectra of EC8, to the values a_g for horizontal earthquakes, and to a_{vg} for vertical earthquakes. Caution: the latter depends on the type of earthquake: 1 or 2.

Functions

This command is used to define a horizontal or vertical spectrum that can be used to study the response of the structure to an earthquake.

This type of seismic response spectrum can be provided point by point, by associating a value of displacement or acceleration with each period value.

We can also call on the standardized spectra in AFPS 92 (elastic spectrum, option AFPS92_E, or calculation spectrum, option AFPS92_C), or in EC 8-1,3.2 (elastic spectrum, option EC8_DF_E, or calculation spectrum, option EC8_DF_C). A standardized spectrum can be defined by its site or its typified parameters, and its adjustment factor. The parameters provided or calculated according to the site are included in the results files. In the Eurocodes, the indexes 1 and 2 refer to the type 1 and 2 earthquakes defined in paragraph 3.2.2.2.

The horizontal and vertical components of a spectrum can be defined independently (it is possible to combine a table of values with a standardized form).

The structure's response for a given period is based on an interpolation or a linear extrapolation of the closest known values.

The nomenclature of the sites is broken down as follows:

- 'S' designates sites defined by the Eurocodes. 'F' designates sites defined by the French decree published on October 26, 2011, on the classification and the rules of paraseismic construction applicable to bridges in the so-called "normal risk" class.
- A,B,C,D,E: designate the class of the ground.
- 1, 2: designates the type of earthquake being studied.

Conditions of use

- ☐ This command must precede the REPONSE MODALE SEISME command that refers to it.
- ☐ All new definitions of the horizontal or vertical spectrum replace the old definitions.
- ☐ For standardized vertical spectra, all the reductive coefficients in the standards are applied automatically.

Examples

```
$ spectrum of vertical seismic response in acceleration defined point/point
SPECTRE SISMIQUE VERTICAL ’SPECTRE VERTICAL D’’ACCELERATION NON NORMALISE’
  ACCELERATION
$   T       Gamma
  0.00    2.00
  1.00    4.00
  3.00    6.00;

$ spectrum of normalized horizontal seismic response
$ defined by its typified parameters
SPECTRE  SISMIQUE  HORIZONTAL
```

```
'SPECTRE HORIZONTAL NORMALISE ELASTIQUE DE L''AFPS92'
NORMALISE  AFPS92_E
PARAMETRES  TB 0.15  TC 0.30  TD 2.07  RA 1.00  RM 2.50
CALAGE 1.50;
```

Related commands

DYNAMIQUE; REPONSE MODALE SEISME

11.17 - EARTHQUAKE MODAL RESPONSE

$$\text{REPONSE MODALE SEISME titre\_reponse COMPOSANTES}\ {}^{*}\begin{Bmatrix}\text{X}\\ \text{Y}\\ \text{Z}\end{Bmatrix}$$

$$\left[\ {}^{*}\left\{\begin{array}{l}\text{ORIENTATION\_SEISME}\quad \theta_z\\ \text{Q}\quad \text{c\_comp}\\ \text{SUPERPOSITION}\quad \left\{\begin{array}{c}\underline{\text{CQC}}\\ \text{SRSS}\end{array}\right\}\\ \text{AMORTISSEMENT}\quad \text{STRUCTURAL}\quad \text{CALCULE}\\ \text{EDITER}\\ \text{ENREGISTRER}\quad \text{no\_reponse}\\ \text{CHARGEMENT}\quad \left\{\begin{array}{l}\text{MODE}\quad \text{no\_mode}\\ \text{EXTREME}\quad \left\{\begin{array}{l}\text{NOEUD}\quad \text{no\_noeud}\\ \text{ELEMENT}\quad \text{no\_element}\end{array}\right\}\\ \qquad \text{COMPOSANTE}\quad \text{no\_comp}\end{array}\right\}\ \text{FACTEUR}\quad \text{facteur}\\ \quad [\text{STRUCTURE}\quad \text{nom\_structure}]\end{array}\right\}\right];$$

Parameters

- titre_reponse: title given to the modal response (character string).
- θ_z: the angle that directs the $OX_sY_sZ_s$ application reference frame of the earthquake by rotation around the OZ axis of the global reference frame. By default, this angle is zero and the earthquake is applied in the global reference frame.
- c_comp: coefficient of behavior (1.0 by default).
- no_reponse: the possible record number of the (total) response. Positive and less than 90_000.
- no_mode: order number of the mode used to generate the possible "mode" nodal load, which is its equivalent, for the PH3 module.
- no_noeud, no_comp: number of the node and number of the component of displacement used to generate the possible "extreme effect" nodal load for the PH3 module.
- no_element, no_comp: number of the element and number of the force component (1 to 6 for the origin and 7 to 12 for the end) used to generate the possible "extreme effect" nodal load for the PH3 module.
- facteur: weighting coefficient to be applied to the load of the mode or extreme effect.
- nom_structure: name of the saved structure to which the load generated for the PH3 module is applied (used on the PHASES SUITE command).

Functions

This command starts the calculation of the response of the structure when subjected to an earthquake, which is predefined and, therefore, provides the orientation, by modal superimposition and spectral analysis.

With the SUPERPOSITION SRSS option, the superimposition of the effects of the selected modes is obtained by simple quadratic accumulation, according to the "SRSS" method. The standard deviations, σ_s, of the resulting effects are obtained by the relation 11.22, with the same ratings

With the SUPERPOSITION CQC option (selected by default), the effects of the selected modes are superimposed by a so-called "CQC" calculation that uses the coefficients of correlation between the modes and effects of the earthquake. The standard deviations are obtained by the relation 11.23, with the same ratings, and in which C_{ij} is calculated according to the conventional formula of the CQC.

Methods of modal superposition used in this module are based on the section 5.2.4 of the book *Calcul dynamique des structures en zone sismique*, Alain CAPRA and Victor DAVIDOVICI, Editions Eyrolles.

With the AMORTISSEMENT STRUCTURAL CALCULE option, the structural damping coefficient of each mode is calculated from the damping characteristics associated with the elements by the AMORTISSEMENT STRUCTURAL ELEMENTAIRE command. By default, AMORTISSEMENT STRUCTURAL MODAL is used.

When the COMPOSANTES option calls several components, the effects they produce are combined quadratically.

If a behavior coefficient other than 1.0 is associated with a standardized calculation spectrum, it acts as a divider of amplitude of the seismic response.

The (total) response can be edited with the EDITER option, and/or recorded in the database, using the ENREGISTRER option. By default, no responses are edited or recorded.

For a structure produced by a non-linear calculation, the forces in the elements are calculated from their matrices of tangential rigidity.

The tangential and normal stresses are calculated on the basis of the elastic behavior of the materials. In cases of mechanical non-linearity, the normal stresses in the corresponding elements are impacted by an infinitely high coefficient and cannot be used.

The standard deviations of the displacements, reactions, force and stresses are always edited, and for each of the components of the corresponding load, the concomitant values of the associated components are calculated according to the relations 11.24, with the same ratings.

For each mode and each of the studied components, the standard deviation and the covariance of the generalized coordinates are also always edited.

With the CHARGEMENT option, the nodal load equivalent to one of the following two types of effects can be generated (in the form of commands for the PH3 module that are stacked in the export file dynph3.don):

- with the MODE option, the nodal load equivalent to the displacements caused by the specified mode is calculated and weighted by the coefficient provided. The nodal forces vector is expressed by:

$$\left\{ F_{mode} \right\} = facteur \cdot \sigma_{mode} \cdot A_{mode} \cdot \left[K \right] \cdot \left\{ X_{mode} \right\} \tag{11.28}$$

- with the EXTREME option, the nodal load equivalent to the so-called "extreme" displacements vector is calculated for all the selected modes, weighted by the coefficient provided. The nodal forces vector is expressed by:

$$\left\{ F_{extreme} \right\} = facteur \cdot \left[K \right] \cdot \left\{ X_{extreme} \right\} \tag{11.29}$$

For the CHARGEMENT EXTREME option, the displacements vector is made up of the standard deviation of the displacement of the specified node, according to the selected component, and the concomitant displacements of the other degrees of freedom. These values correspond to the total effect of the earthquake (see 11.27).

If no STRUCTURE is specified, the current STRUCTURE is used in the PHASE SUITE command of the generated loading.

If a STRUCTURE is specified, nom_structure is used in the PHASE SUITE command of the generated loading. The current STRUCTURE is then used to again save the STRUCTURE in the generated loading. This can then be reused for a dynamic calculation. It is then possible to form a series of dynamic calculations and loadings, in which the current structure is updated according to the loading applied.

Conditions of use

- Several REPONSE MODALE commands can be introduced in the same session.
- This command must be preceded by the MODES, AMORTISSEMENT STRUCTURAL and SPECTRE commands (for the horizontal and/or vertical directions) and, possibly, by the SELECTION MODES and ADMITTANCE MODALE commands.
- The identification number of an EFFET already recorded by the PH3, ENV, DYN or ETU modules (according to the terminology used by the ETU module, see Table 12.1) cannot be used to designate a recorded modal REPONSE.
- If the earthquake impacts the X or Y component, a horizontal spectrum must be defined.
- If the earthquake impacts the Z component, a vertical spectrum must be defined.
- The coefficient of behavior is only compatible with the calculation spectra.

Methodological advice

- The nodal mode or extreme force loadings are reserved for the non-linear calculations.
- To make correct use of the EXTREME or MODE options with a reference STRUCTURE, you must first make a double backup in PH3. One of the two backups will appear in the STRUCTURE command of the DYN module and saved STRUCTURE (SAUVE nom_structure) in the generated loading. The other will appear in the CHARGEMENT command and will be used as a reference STRUCTURE for the generated loading (PHASE SUITE). The reference STRUCTURE remains unchanged from one calculation to another, while the STRUCTURE used for the dynamic calculation is updated. The data sequence is as follows:

```
$ initial phasing
SAUVE structref
SAUVE structdyn
FIN
$ current step
DYNAMIQUE;
STRUCTURE structdyn;
REPONSE  MODALE  VENT
…
CHARGEMENT EXTREME … STRUCTURE structref
…
FIN
$ extreme loading in the file dynph3.don
PHASAGE SUITE structref
CAS DE CHARGE
```

```
          SAUVE structdyn
          FIN
```

Examples

```
$ ----------------------------------------------------------------------
$ modal response of the structure to a horizontal earthquake at 45.0
$ degrees relative to the global OX axis, of which component X is studied
$ the response calculated using the CQC method (used by default) is edited
$ and saved with the number 100; no loading generated for PH3
$ ----------------------------------------------------------------------
REPONSE  MODALE  SEISME
    'SEISME HORIZONTAL A 45.0 DEGRES'
    COMPOSANTES  X
    ORIENTATION_SEISME  45.0
    EDITER
    ENREGISTRER  100;

$ ----------------------------------------------------------------------
$ modal response of the structure to a global earthquake, of which the
$ three
$ components in global coordinates are studied (no orientation defined by
$ default)
$ the response calculated in quadratic superposition is saved with
$ number 200 and edited; the nodal loading of extreme effect
$ calculated for node 1000 and the component of displacement 3 (Uz) is
$ weighted by coefficient 2.0 and translated into commands for module PH3
$ which will use the structure STRUC_2 in association
$ ----------------------------------------------------------------------
REPONSE  MODALE  SEISME
    'SEISME MULTIDIRECTIONNEL'
    COMPOSANTES  X  Y  Z
    SUPERPOSITION  SRSS
    EDITER
    ENREGISTRER  200
    CHARGEMENT  EXTREME  NOEUD  1000  COMPOSANTE  3  FACTEUR  2.0
    STRUCTURE  STRUC_2;
```

Related commands

DYNAMIQUE ; MODES ; SELECTION MODES

AMORTISSEMENT STRUCTURAL; ADMITTANCE MODALE; SPECTRE

11.18 - DETERMINISTIC ACTION

ACTION DETERMINISTE titre_action

$$
\left[\ \ast \left\{ \begin{array}{l}
\text{INTENSITE} \quad \text{FOURIER} \quad T \quad a_o \quad \langle a_i \quad b_i \rangle_n \\
\text{INTENSITE} \quad \text{MODALE TOTALE} \quad \text{no\_mode [FREQUENCE f]} \\
\text{INTENSITE} \quad \text{MODALE PARTIELLE} \quad \text{no\_mode [FREQUENCE f]} \\
\text{INTENSITE} \quad \text{TABULEE} \quad \langle t_i \quad v_i \rangle_n \\
\text{INTENSITE} \quad \text{FICHIER} \quad \text{fichier} \\
\text{MOBILE} \quad \text{VITESSE} \quad \text{vitesse} \quad \text{ACCELERATION} \quad \text{accélération} \\
\text{DUREE} \quad \text{DEBUT} \quad \text{t\_deb} \quad \text{[FIN} \quad \text{t\_fin]}
\end{array} \right\} \right]
$$

$$
\ast\ast \left\{ \begin{array}{l}
\text{POIDS} \quad \text{CAS} \quad \text{no\_cas} \\
\text{FORCE} \quad \text{CAS} \quad \text{no\_cas} \\
\text{POIDS} \quad \text{POUTRE} \quad \text{no\_ptre} \quad \text{COMPO ic VALEUR v POSITION x y z} \\
\text{FORCE} \quad \text{POUTRE} \quad \text{no\_ptre} \quad \text{COMPO ic VALEUR v POSITION x y z} \\
\text{MASSE} \quad \text{POUTRE} \quad \text{no\_ptre} \quad \text{COMPO ic VALEUR v POSITION x y z} \\
\text{POIDS} \quad \text{SUPPORT} \quad \text{no\_sup} \quad \text{COMPO ic VALEUR v POSITION x y z} \\
\text{FORCE} \quad \text{SUPPORT} \quad \text{no\_sup} \quad \text{COMPO ic VALEUR v POSITION x y z} \\
\text{MASSE} \quad \text{SUPPORT} \quad \text{no\_sup} \quad \text{COMPO ic VALEUR v POSITION x y z} \\
\text{POIDS} \quad \text{NOEUD} \quad \text{no\_noeud} \quad \text{COMPO ic VALEUR v} \\
\text{FORCE} \quad \text{NOEUD} \quad \text{no\_noeud} \quad \text{COMPO ic VALEUR v} \\
\text{MASSE} \quad \text{NOEUD} \quad \text{no\_noeud} \quad \text{COMPO ic VALEUR v} \\
\text{ACCELERATION} \quad \text{NOEUD} \quad \text{no\_noeud} \quad \text{COMPO ic} \quad \text{VALEUR v} \\
\text{DEPLACEMENT} \quad \text{NOEUD} \quad \text{no\_noeud} \quad \text{COMPO ic} \quad \text{VALEUR v}
\end{array} \right\} ;
$$

Parameters

- titre_action: title given to the deterministic action (character string).
- T: period of pulsation of the Fourier series in seconds.
- a_o: constant term of the Fourier series.
- a_i: ith cosine coefficient of the Fourier series.
- b_i: ith sine coefficient of the Fourier series.
- t_i,v_i: table defining the parameters of temporal evolution.
- fichier: name of the file containing the tabulated values t_i,v_i.
- vitesse: value of the speed of the action.
- accElEration: value of the acceleration of the action.
- t_deb: instant in seconds of the start of the action (0 seconds or t_o by default).
- t_fin: instant in seconds of the end of the action (without end or t_n by default).
- no_cas: number of the PH3 load case to be applied.
- no_ptre: number of the beam to be loaded.
- no_sup: number of the ENV support to be loaded.
- no_noeud: number of the node of the model to be loaded.
- ic: number of the loading component.
- v: intensity of the action.

- x,y,z: initial loading coordinates.

Functions

This command is used to define a deterministic dynamic action. This action is automatically applied when processing the REPONSE TEMPORELLE command that follows this command.

The INTENSITE, MOBILE and DUREE options can be used to check the application of the load. They are optional and are not mutually exclusive. By default, the dynamic load is fixed, of constant intensity and is applied from the start of the instant in question until the end.

With the INTENSITE FOURIER option, the intensity of the load is modulated by a Fourier series that is defined as follows:

$$facteur = a_0 + \sum_{i=1,n} \; a_i \cos(2\,\pi\, i\, \frac{t}{T}) + b_i \sin(2\,\pi\, i\, \frac{t}{T})$$

The factor obtained at time t, which is counted from the start of the deterministic action t_deb, is applied to the intensity of the load.

With the INTENSITE MODALE TOTALE option, the load intensity is modulated by a given f frequency sinusoidal function or otherwise that of the nu_mode mode:

$$facteur \;=\; sin(2\;\pi\;\; f)$$

The factor obtained at t time, counted since the beginning of the deterministic action: t_deb, is applied to the intensity of the load. If the direction of the initial force (before the factor is applied) to a node is the opposite (in the sense of the scalar product) of that of the nodal displacement of the selected mode, the nodal charge sign is **reversed.** This option applies only to actions of the non-MOBILE FORCE type. It corresponds to the pedestrian loads as indicated in the 2006 Sétra Methodological Guide: " Passerelles piétonnes -Évaluation du comportement vibratoire sous l’action des piétons " (Pedestrian Bridges - Assessment of Vibrational Behaviour Under Pedestrian Action).

With the INTENSITE MODALE PARTIELLE, the load intensity is modulated by a given f frequency sinusoidal function or otherwise that of the nu_mode mode:

$$facteur \;=\; sin(2\;\pi\;\; f)$$

The factor obtained at t time, counted since the beginning of the deterministic action: t_deb, is applied to the intensity of the load. If the direction of the initial force (before the factor is applied) to a node is the opposite (in the sense of the scalar product) of that of the nodal displacement of the selected mode, the nodal charge is **cancelled.** This option applies only to actions of the non-MOBILE FORCE type. It corresponds to pedestrian loads that are applied only if they have an adverse effect on the selected mode. If they are favorable they are cancelled.

With the INTENSITE TABULEE or the INENSITE FICHIER option, the intensity of the load is modulated by the value obtained at the time t by linear interpolation of the tabulation values t_i ,vi.

$$facteur = v_{i-1} + (\frac{v_i - v_{i-1}}{t_i - t_{i-1}})(t - t_{i-1}) \quad pour\, t \in \left[t_{i-1}, t_i\right]$$

$$facteur = v_1 \quad pour \quad t < t_1$$

$$facteur = v_n \quad pour \quad t > t_n$$

The time t is the time of the temporal response. The factor obtained at time t is applied to the intensity of the load. The INTENSITE options are forbidden for POIDS and MASSES.

With the MOBILE option, a speed and/or an acceleration are assigned to the action. This option is forbidden with CAS DE CHARGE and actions on the NŒUDS. The origin of the time is taken to equal 0 by default, otherwise as equal to t_deb. The initial position is the position indicated by the definition of the action. The positive direction is the direction of travel of the SUPPORT or the POUTRE.

With the DUREE option, an instant of the start of the application t_deb is assigned to the action and, optionally, an instant of the end of the application, t_fin. The origin of the time is zero. By default, the start and the end of the application of the action are the same as those of the complete temporal range defined by the ACTION TEMPORELLE command.

The actions may be of one of the following types:

- CAS: a numbered CAS DE CHARGE of PH3 is applied. If the POIDS option is indicated, the applied forces and the corresponding masses are considered. If the FORCE option is indicated, only the forces of the loading case are applied.
- POUTRE: a FORCE or a POIDS (MASSE+FORCE) of intensity v is applied according to the component ic at the position x, y and z of the POUTRE in the reference frame of the FIBRE REPERE.
- SUPPORT: a FORCE or a POIDS (MASSE+FORCE) of intensity v is applied according to the component ic at the position x, y and z of the SUPPORT, defined in ENV data. This support must be a simple support made up of elements or beams. It must not be made up of supports or nodes.
- NŒUD: a FORCE, a MASSE, a POIDS, an ACCELERATION or a DEPLACEMENT, of intensity v, is applied according to the component ic to the node no_noeud.

These different loadings can be combined with one another. A MASSE or a POIDS can only be applied to the vertical component. Their intensity is always positive. An ACCELERATION or a DEPLACEMENT can only be applied to a support. The indicated component is the component according to the global reference frame, not the local reference frame.

Conditions of use

- Several ACTIONS DETERMINISTES can precede the REPONSE TEMPORELLE command(s) that process them.

Examples

```
$ ---------------------------------------------------------------------------
$ deterministic action representing a vehicle impact defined in the form of
$ load case number 100 of PH3. The mass and the force exerted are
$ considered in the temporal analysis. The shock is applied at moment 0.
$ Only the vertical component of the loading is used to determine
$ the mass of the truck.
$ ---------------------------------------------------------------------------
ACTION DETERMINISTE
    'CHOC D UN CAMION SUR UNE PILE'
    POIDS CAS 100   ;

$ ---------------------------------------------------------------------------
$ deterministic action representing the displacement of a 250-ton vehicle
$ on a deck, traveling at a speed of 10 m/s and whose initial position
$ is at the origin of the deck and at the ordinate: -3.00 m.
```

```
$ -------------------------------------------------------------------------
ACTION DETERMINISTE
    'DEPLACEMENT D UN CAMION SUR UN TABLIER'
    MOBILE VITESSE 10
    POIDS POUTRE 1 COMPOSANTE 3 VALEUR 250 POSITION 0. -3.00 0.20 ;

$ -------------------------------------------------------------------------
$ deterministic action representing the displacement of a 250-ton vehicle
$ on a support made of a succession of elements, traveling at a speed
$ of 10 m/s, and accelerating at 2 m/s/s, and whose initial position
$ is at the origin of the support and at the ordinate: -3.00 m.
$ -------------------------------------------------------------------------
ACTION DETERMINISTE
    'DEPLACEMENT D UN CAMION SUR UNE SUCCESSION D ELEMENTS'
    MOBILE VITESSE 10 ACCELERATION 2
    POIDS SUPPORT 1 COMPOSANTE 3 VALEUR 250 POSITION 0. -3.00 0.20 ;
```

```
$ ---------------------------------------------------------------------------
$ deterministic action representing a horizontal sinusoidal harmonic
$ action of an intensity of 2.5 tons, a period of 1.5 seconds, a mean value
$ of 2 and a factor of 4, starting 10 seconds from the original time and
$ ending 100 seconds from the original time;
$ ---------------------------------------------------------------------------
ACTION DETERMINISTE
    'FORCE SINUSOIDALE HORIZONTALE '
    INTENSITE FOURIER 1.5 2. 0. 4.
    DUREE DEBUT 10 FIN 100
    FORCE NOEUD 20  COMPOSANTE 2 VALEUR 2.5 ;

$ deterministic action representing a horizontal sinusoidal harmonic
$ action of an intensity of 2.5 tons, a period of 1.5 seconds, a mean value
$ of 2 and a factor of 4, moving on a beam at a speed of 5 m/s;
$ ---------------------------------------------------------------------------
ACTION DETERMINISTE
    'FORCE SINUSOIDALE HORIZONTALE MOBILE'
    MOBILE VITESSE 5
    INTENSITE FOURIER 1.5 2. 0. 4.
    FORCE POUTRE 10  COMPOSANTE 2 VALEUR 2.5 POSITION 0.0 0.0 0.0 ;

$ ---------------------------------------------------------------------------
$ deterministic actions representing an accelerogram, horizontal axis Y
$ and vertical s exerted on the supports at nodes 10, 20 and 30.
$ ---------------------------------------------------------------------------
ACTION DETERMINISTE
    'ACCELOGRAMME HORIZONTAL AXE Y'
    INTENSITE TABULEE 0. 1. 0.01 2…
    ACCELERATION NOEUD 10  COMPOSANTE 2 VALEUR 1.00
    ACCELERATION NOEUD 20  COMPOSANTE 2 VALEUR 1.00
    ACCELERATION NOEUD 30  COMPOSANTE 2 VALEUR 1.00 ;
ACTION DETERMINISTE
    'ACCELOGRAMME VERTICAL'
    INTENSITE TABULEE 0. 1.5. 0.01 1.4…
    ACCELERATION NOEUD 10  COMPOSANTE 3 VALEUR 1.00
    ACCELERATION NOEUD 20  COMPOSANTE 3 VALEUR 1.00
    ACCELERATION NOEUD 30  COMPOSANTE 3 VALEUR 1.00 ;
```

Related commands

DYNAMIQUE; REPONSE TEMPORELLE; CAS DE CHARGE (PH3)

SUPPORT (ENV)

11.19 - ACCIDENTAL ACTION

ACTION ACCIDENTELLE titre_action

$$\left[\;^{**}\left\{\begin{array}{lll} \text{RUPTURE} & \text{ELEMENT} & \langle \text{no\_element} \rangle_n \\ \text{RUPTURE} & \text{APPUI} & \langle \text{no\_noeud} \rangle_n \\ \text{RUPTURE} & \text{CABLE} & \langle \text{nom\_cable} \rangle_n \end{array}\right\}\right];$$

Parameters

- titre_action: title given to the accidental action (character string).
- no_element: number of the deleted element.
- no_noeud: number of the node bearing a support to be deleted.
- nom_câble: name of the deleted cable.

Functions

This command is used to define an accidental dynamic action. This action is automatically applied at the time t_debut when processing the REPONSE TEMPORELLE command that follows this command.

Conditions of use

- This command must be introduced before the REPONSE TEMPORELLE command that processes it.
- The different deleted entities must be distinct.
- This command can be combined with a deterministic action.

Examples

```
$ -------------------------------------------------------------------
$ accidental simultaneous rupture action of elements and cables.
$ -------------------------------------------------------------------
ACTION ACCIDENTELLE
    'RUPTURE DE CABLES ET D ELEMENTS’
    RUPTURE ELEMENTS  10 11 12
    RUPTURE CABLES CABLE21 CABLE22 CABLE23;

$ -------------------------------------------------------------------
$ accidental simultaneous rupture action of several supports.
$ -------------------------------------------------------------------
ACTION ACCIDENTELLE
    'RUPTURE D APPUIS’
    RUPTURE APPUIS  10 20 30;
```

Related commands

DYNAMIQUE; REPONSE TEMPORELLE

11.20 - STOCHASTIC WIND ACTION

ACTION STOCHASTIQUE VENT titre_action

PERIODE To_r

ECHANTILLONS_FREQUENTIELS nb_frequences

$$\left[{}^{*}\left\{ \begin{array}{l} \text{AMORTISSEMENT\_AERODYNAMIQUE} \\ \text{DISTANCE\_CORRELATION} \quad \text{d\_correlation} \\ \text{ORIENTATION\_VENT} \quad \theta_z \quad \theta_y \\ \text{DENSITES\_SPECTRALES} \quad \text{NOEUDS} \quad \left\langle \text{no\_noeud\_1} \quad \text{no\_noeud\_2} \right\rangle_n \\ \qquad \left(\left\{ \begin{array}{l} \text{FENETRE} \quad \left\{ \begin{array}{l} \underline{\text{PARZEN}} \quad \text{largeur} \\ \text{PERIODOGRAMME} \quad \text{nb\_f\_moy} \end{array} \right\} \\ \text{INTEGRATION} \end{array} \right\} \right) \\ \text{SIMULATION} \quad \text{SEMENCE} \quad \text{no\_semence} \quad \text{SERIES} \quad \text{nb\_series} \end{array} \right\} \right];$$

Parameters

- titre_action: title given to the stochastic wind action (character string).
- To_r: reference period of the generated wind.
- nb_frEquences: the number of reference frequency samples. Must be a power of 2 greater than 8 and less than or equal to 32,768.
- no_nœud_1 and no_nœud_2: numbers of the density calculation nodes.
- d_correlation: maximum distance to take the correlation into consideration.
- θ_z, θ_y: the angles that orientate the wind reference frame by rotation around the OZ axis, then around the "new" OY axis, of the global reference frame. By default, these angles are zero, the Ou, Ov and Ow axes of the wind reference frame are respectively the same as the OX, OY and OZ axes of the global reference frame and the mean speed vector is parallel to OX (Figure 11.5).
- largeur: width between 1 and 4 of the window to be considered for the truncation (3 by default).
- nb_f_moy: number of averaging frequencies of the periodogram (recommended value: 10);
- no_semence: number of the stochastic seeding to be considered for the simulation (must be greater than 0).
- nb_sEries: number of temporal series to be generated for the simulation.

Functions

This command is used to define a stochastic wind type dynamic action. This action is automatically applied when processing the REPONSE TEMPORELLE command that follows this command.

The period To_r is used to determine the longest period of the generated synthetic wind. The scope of the spectrum is defined by:

$$f \in \left[\frac{1}{T_{o\_r}}, \frac{n_f}{2T_{o\_r}} \right]$$

Here n_f is the nb_frequences parameter. The time interval of the temporal discretization equals:

$$\Delta t = \frac{T_{o\_r}}{n_f}$$

The time interval of the frequency discretization equals:

$$\Delta f = \frac{1}{T_{o\_r}}$$

The scope of the frequency spectrum is an essential parameter of the quality of the analysis.

With the AMORTISSEMENT_AERODYNAMIQUE option, the speed of the wind is corrected by the speed of the structure to obtain an apparent wind speed that is used to calculate the forces on the structure.

With the DISTANCE_CORRELATION option, the distance is fixed, beyond which the interspectral densities are assumed to be zero. This parameter is used to reduce the size of the matrices of interspectral density. However, this parameter must be used with care because if it is too small, it can result in matrices of interspectral density that are not defined as positive. The default value is 1,000 m.

With the DENSITES_SPECTRALE option, it is possible to ask for the calculation of the interspectral densities of the generated wind of the nodes no_nœud_1 and no_nœud_2 in question, and of the corresponding theoretical values. If the two nodes have the same number, the spectral density of the wind at the node in question is analyzed. These elements are edited, either using the SIMULATION option below, or in the TEMPOREL calculation itself.

The FENETRE is used to indicate the mode of calculation of the spectral density. With the PARZEN option, a PARZEN apodization window is applied. The width of this window is controlled by the width parameter, which is 1 for the smallest and 4 for the widest. The recommended value is 3. With the PERIODOGRAMME option, the periodogram of the speed is calculated directly and averaged by taking nb_freq values on either side of the calculation value. This option does not work for interspectral densities (a zero value is returned). With the INTEGRATION option, the auto-correlation is calculated and integrated to obtain the spectral density. Note that the precision of the calculation of the spectral density is less good on low frequencies and excellent on high frequencies. It is possible that an estimate of the spectral density is obtained for low frequencies, to within a factor of 2.

With the SIMULATION option, it is possible to obtain an estimate of the densities of the wind that will be generated, before the effective calculation of the effects of the wind. In this case, it is necessary to indicate the number of the random generation seeding of the phase shifts and the number of series to be generated, given that a series lasts To_r seconds. This option is only meaningful if the DENSITES_SPECTRALES option is encoded.

The general method for the generation of the wind forces for each series is as follows:

- Calculation of the matrices of interspectral density of the nodes impacted by the wind for each sampling frequency and in each wind direction,
- Factorization of these matrices,
- Generation, for each node that is not impacted by the wind and for each frequency, of a list of phase shifts,
- Calculation of the frequency amplitudes of the speeds in each node using the matrices of spectral density,
- Calculation of the temporal amplitudes of the speeds for each node by inverted Fourier transformation according to the FAST FOURIER TRANSFORMATION method,
- For each time interval, calculation of the current speed of the wind in each node and in the three directions,
- Calculation of the relative speed using an option,
- Calculation of the forces in the nodes, using the aerodynamic coefficients of the elements.

In a SIMULATION or a TEMPOREL calculation, the following control elements are produced for each pair of nodes of the DENSITES_SPECTRALES option:

- Table of the spectral and interspectral densities for each sampling frequency,
- Average deviation between the densities of the simulated speeds and the specified density,
- Calculated standard deviations on the basis of the initial spectral density and on the basis of the spectral density of the simulated wind,
- Mean speed vector and mean forces vector,
- Mean global deviation between the specified spectral densities and the simulated densities of the wind for all the control nodes.

In the presence of winds with different characteristics, the interspectral densities are calculated by taking a zero interspectral density into consideration for two nodes located in two distinct wind zones.

Note that the generated phase shifts are identical from one calculation session to another, provided that the seeding number is the same. The seeding defines a reproducible temporal scenario, because it initializes the "random" function used to generate the random signals. For two successive calculations in two distinct sessions to be independent, the difference between the two seedings must be at least equal to 3*nnoe*n_f*nb_sEries, where nnoe is the number of nodes impacted by the wind.

The forces on the structure are condensed at the nodes, but they are calculated using the aerodynamic characteristics of the elements.

Conditions of use

- This command must be introduced before the REPONSE TEMPORELLE command that processes it.
- One or more winds must be defined beforehand.
- If several winds are defined, they must be geometrically separate, i.e. there is no overlap.
- A structure must be specified beforehand.
- This command can be combined with a deterministic action.

Methodological advice

- It is advisable to run a simulation to verify that the sampling parameters are correct. The densities will be chosen on meaningful nodes of the structure with regard to the wind.
- The number of temporal series must always be 5 or more. For a number of series equal to or greater than 20, the mean results are quite close to the asymptotic value. It is also necessary to check that, on average, the generated system of forces is close to the effects of the mean wind. The control elements produced by PCP will be analyzed in this respect.
- The reference period must be long enough to represent the low frequencies. Moreover, it must be long relative to the period of the structure's first mode of vibration. It is advisable not to drop below 256 seconds and to take a power of 2 as the value to obtain a simple time interval. Note that this period determines the frequency discretization interval. The longer the period, the smaller the frequency interval and the finer the discretization.
- The time interval of the temporal sampling must be greater than or equal to the discretization interval of the temporal analysis.
- The number of sampling frequencies must be such that the high frequencies are sufficiently represented. The resulting range of frequencies must contain the significant modes of vibration of the structure.

Examples

```
$ -----------------------------------------------------------------------
$ wind blowing in Y. the densities are calculated at nodes 200 and 201.
$ aerodynamic damping is active. the number of frequency samples
$ is taken to equal 512 and the reference period is 25.6s
$ the time graduation used for the simulation is 0.025 seconds , the
$ tack is taken to equal 1. the number of temporal series is set
$ at 20.
$ -----------------------------------------------------------------------
ACTION STOCHASTIQUE VENT
    " VENT DE NORD-OUEST STOCHASTIQUE"
    PERIODE           25.6
    FREQUENCES        512
    ORIENTATION_VENT 90 0
    AMORTISSEMENT AERODYNAMIQUE
    DENSITES_SPECTRALES NOEUDS 200 200 201 201 200 201
    SIMULATION SEMENCE 1 SERIES 20;
```

Related commands

DYNAMIQUE ; STRUCTURE ; VENT ; REPONSE TEMPORELLE

11.21 - TEMPORAL RESPONSE

REPONSE TEMPORELLE titre_reponse

$$[\text{AMORTISSEMENT STRUCTURAL} \left\{\begin{array}{llll} \text{RAYLEIGH} & & a & b \\ \text{RAYLEIGH} & \text{CALCULE} & f_1 & f_2 \\ \text{MODAL} & & a & b \\ \text{MODAL} & \text{CALCULE} & a & b \\ \text{MODAL} & \text{FILTRE} & f_c & d_c \end{array}\right\}]$$

$$[\text{AMORTISSEMENT} \quad \text{DIRECT} \left\{\begin{array}{l} \underline{\text{TANGENT}} \\ \text{MIXTE } f_{\text{tangent}} \ f_{\text{secant}} \ \text{nb\_ite} \end{array}\right\} (\text{vitesse\_troncature})]$$

$$[\ {}^{*}\left\{\begin{array}{ll} \text{MATRICE} \quad \text{AMORTISSEMENT} & \left\{\begin{array}{l} \text{IGNOREE} \\ \text{INITIALE} \\ \underline{\text{TEMPORELLE}} \\ \text{EXACTE} \end{array}\right\} \\ \text{MATRICE} \quad \text{RIGIDITE} & \left\{\begin{array}{l} \text{TANGENTE} \\ \underline{\text{EXACTE}} \end{array}\right\} \\ \text{MATRICE} \quad \text{MASSE} & \left\{\begin{array}{l} \text{IGNOREE} \\ \text{INITIALE} \\ \underline{\text{TEMPORELLE}} \\ \text{EXACTE} \end{array}\right\} \end{array}\right\}]$$

$$[\text{NEWMARK} \quad p_1 \quad p_2] \ [\text{SEMENCE} \quad \text{no\_semence}]$$

$$[\text{INDICES} \quad \text{indice\_deb} \quad \text{indice\_fin}]$$

$$[\text{CINEMATIQUE} \left\{\begin{array}{l} \underline{\text{ABSOLUE}} \\ \text{RELATIVE} \end{array}\right\}]$$

$$[\ {}^{*}\left\{\begin{array}{l} \text{ETAT} \quad [\text{EDITER} \ [\text{options\_editions}]] \quad [\text{ENREGISTRER no\_etats}] \quad \text{PAS pas\_e} \\ \text{EXTREMAS} \quad [\text{EDITER}] \quad [\text{ENREGISTRER no\_reponse}] \quad [\text{options\_extremas}] \end{array}\right\}]$$

[options non lineaires]

DEBUT t_debut FIN t_fin PAS t_pas ;

$$[\text{options\_editions}] = [\text{options\_extremas}] = [\ {}^{*}\left\{\begin{array}{ll} \text{ACCELERATIONS} & \\ \text{VITESSES} & \\ \text{DEPLACEMENTS} & \\ \text{REACTIONS} & \\ \text{EFFORTS} & \text{ELEMENTS} \\ \text{EFFORTS} & \text{SECTIONS} \\ \text{CONTRAINTES NORMALES} & \\ \text{CONTRAINTES TANGENTES} & \end{array}\right\}]$$

Parameters

- titre_reponse: title given to the temporal response (character string).
- a and b: RAYLEIGH coefficients for the construction of the damping matrix.
- f_1 and f_2: frequency of calculation of the RAYLEIGH coefficients.
- f_c: frequency of filtering of the superior modes.
- d_c: increment expressed as a percentage of the critical damping of the damping beyond the filtering frequency f_c.
- $f_{tangent}$ and f_{secant}: factors of the tangential and secant damping in MIXTE mode.
- nb_ite: number of equilibrium iterations, above which the MIXTE mode is used.
- vitesse_troncature: speed below which the power law of the damping is degenerated into a cubic law tangential to the real curve at the speed of truncation. Its default value is 1.e-06.
- p_1 and p_2: parameters used to control the NEWMARK solution method. By default, both these values are set at 0.5.
- no_semence: number of the stochastic seeding to be considered for a stochastic simulation (1 by default).
- indice_deb and indice_fin: numbers of the time interval at the start and at the end of the temporal range to be generated. By default, the temporal range is complete, therefore indice_deb=1 and indice_fin =(t_fin-t_debut)/tpas+1.
- no_etats: record number of the various successive temporal reports, positive and less than 999. This number is independent of the effects numbers.
- no_reponse: record number of any positive responses, positive and less than 90_000. This number must be compatible with the effects numbers already recorded.
- pas_e: frequency of creation or recording of the successive reports. Only report numbers that are multiples of this frequency are created and/or recorded. 0, pas_e, 2 pas_e, etc.;
- t_debut and t_fin: initial and final times of the temporal calculation in seconds, with t_debut ≥ 0 and t_fin < 9999 ;
- t_pas: time increment in seconds to be applied between two instants of temporal calculation. This is the temporal discretization interval.

Functions

This command starts the calculation of the temporal response to one or several deterministic actions, an accidental action and/or a stochastic action. The only mandatory parameters are the time in seconds of the start and end of the calculation of the temporal response and the temporal discretization interval. The considered structure is the structure defined by the STRUCTURE command.

Note that the different temporal calculation options can be used to inhibit the dynamic effects and to make a purely static calculation, for example, when loads are displaced.

The masses introduced by the user in the form of ACTIONS can be taken into consideration in their current configuration in different ways, which are more or less precise, using the MATRICE MASSES option.

The damping taken into consideration is calculated using different methods, according to the AMORTISSEMENTS ELEMENTAIRES, MODAUX and/or DIRECTS introduced by the user. The user must pay attention to the conditions of the damping, because they determine the structure's response. In non-linear calculations, the plastification damping is not specifically taken into consideration by the software. This means that if the structure is likely to plastify, corresponding additional structural damping must be introduced.

STRUCTURAL DAMPING

By default, no AMORTISSEMENT STRUCTURAL is applied. However, if the user has introduced an AMORTISSEMENT DIRECT, it is automatically taken into consideration, even in the absence of any structural damping.

The following options apply to AMORTISSEMENTS STRUCTURALS:

- RAYLEIGH: Damping is calculated according to the so-called RAYLEIGH method, which consists of building the damping matrix as a combination of the matrix of masses weighted by a, and of the matrix of rigidity, weighted by b.

$$[A]=a\,[M]+b\,[K]$$

- RAYLEIGH CALCULE: Damping is calculated according to the so-called RAYLEIGH method, but the coefficients a and b related to each element or node are calculated from the value of the frequencies f_1 and f_2, and from the AMORTISSEMENTS STRUCTURAUX ELEMENTAIRES ξ_e and ξ_n as follows:

$$[A_e]=a_e\,[M_e]+b_e\,[K_e]$$

$$[A_n]=a_n\,[M_n]+b_n\,[K_n]$$

$$\begin{Bmatrix} a_e \\ b_e \end{Bmatrix}=\frac{2\xi_e}{\omega_1+\omega_2}\begin{Bmatrix} \omega_1\omega_2 \\ 1 \end{Bmatrix} \quad \begin{Bmatrix} a_n \\ b_n \end{Bmatrix}=\frac{2\xi_n}{\omega_1+\omega_2}\begin{Bmatrix} \omega_1\omega_2 \\ 1 \end{Bmatrix}$$

$$\omega_i=2\pi\,f_i$$

$$[A]=\sum[A_e]+\sum[A_n]$$

This option can only be introduced if the user has introduced AMORTISSEMENTS ELEMENTAIRES.

- MODAL: The total damping is the sum total of the damping associated with each mode calculated as a specific participation of the mode in the general damping, and of a RAYLEIGH damping, calculated from the coefficients a and b.

$$[A]=a\,[M]+b\,[K]+[M]\left[\sum_{i=1,n}\frac{2\xi_i\omega_i}{m_i}X_iX_i^T\right][M]$$

The parameters ξ_i, ω_i and m_i are respectively the damping, the pulsations and the generalized masses associated with each mode. This option can only be introduced if a calculation of the MODES was requested previously. If the RAYLEIGH coefficients are different from 0, the user must take this additional damping into consideration by subtracting it when evaluating the modal damping.

MODAL CALCULE: Damping is calculated in the same way as for simple MODAL, but the modal damping is calculated from the characteristics of the materials and the supports. This option can only be introduced if the user has introduced AMORTISSEMENTS ELEMENTAIRES and if a calculation of the MODES was previously requested. If the RAYLEIGH coefficients are different from 0, the user must take this additional damping into consideration by subtracting it when evaluating the modal damping.

- MODAL FILTRE: Damping is calculated in the same way as for simple MODAL CALCULE, but the modal damping is calculated from the characteristics of the materials and the supports. But the RAYLEIGH coefficients are calculated so that the straight line of RAYLEIGH damping (a=0) passes through the point defined by f_c and the minimal modal damping. Beyond f_c and up to the frequency of the last calculated mode f_n, the modal damping is increased by d_c. Beneath f_c, it is diminished by the value of the RAYLEIGH damping.

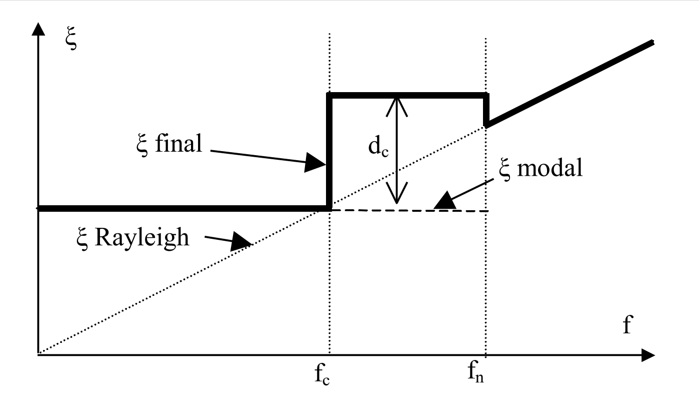


Figure 11.6 – Filtered damping curve

Modal damping in the presence of a mixed structure varies from one mode to another. In this case, the minimal damping is used as the modal damping.

$$[A]=a[M]+b[K]+[M]\left[\sum_{i=1,n}\frac{2\xi_i\omega_i}{m_i}X_iX_i^T\right][M]$$

$$avec \quad a=0 \quad b=\frac{\xi_{\min}}{\pi f_c}$$

$$\xi_i=\xi_i^{ini}-\frac{b}{2}\omega_i \quad si \quad f_i\le f_c$$

$$\xi_i=\min(\xi_i^{ini}+d_c,\xi_{\min}+d_c)-\frac{b}{2}\omega_i \quad et \quad \xi_i>0 \quad si \quad f_i>f_c$$

$$\xi_{\min}:\min_{i=1,n}(\xi_i^{ini})$$

ξ_i^{ini} : Amortissement modal initial avant correction du i^{eme} mode

This option can only be introduced if the user has defined AMORTISSEMENTS ELEMENTAIRES and if a calculation of the MODES was previously requested.

DIRECT DAMPING

This option is used to modify the default behavior of the AMORTISSEURS DIRECTS. By default, the convergence operator of the damping is taken to equal the TANGENT operator. With this option, it can be fixed, on the basis of nb_ite iterations of equilibrium, at a MIXTE value obtained by the linear combination of the tangent operator and the secant operator, using the coefficients $f_{tangent}$ and f_{sEcant}. For the damping power laws, a speed of truncation can be specified in order to degenerate the power law into a cubic law for speed values lower than the truncation speed. The MIXTE operator reduces the risk of oscillation, but slows down the calculations. Increasing the truncation speed also reduces the risk of oscillation. When the origin of an oscillation of the convergence in equilibrium is the direct damping, it is possible to first increase the speed of truncation, then to use the mixed damping with values such as $f_{tangent}+f_{sEcant} = 1$ and $f_{tangent}$ greater than or equal to f_{sEcant}. The recommended values are respectively 0.75 and 0.25. The number of switchover iterations could be 50.

CALCULATION MATRICES

The calculation options of the MATRICES are as follows:

- ☐ RIGIDITE TANGENTE: The matrix of tangential rigidity calculated by PH3 when saving the STRUCTURE is used and remains unchanged throughout the temporal calculation process. All the non-linear calculation options are inhibited. For linear calculations including a chained calculation of the stays, the latter is also inhibited. This option is used to make a linear TEMPOREL calculation further to non-linear construction phasing, or in the presence of ELASTOPLASTIQUES entities.
- ☐ RIGIDITE EXACTE: All the non-linear calculation options of the rigidity of the PH3 module are reproduced for the temporal calculation. The same applies to chained linear calculations. This is the default option.
- ☐ MASSE IGNOREE: The masses matrix is not taken into consideration. This option is used to obtain the purely static effects of a given ACTION.
- ☐ MASSE INITIALE: The masses matrix is calculated once and is not subsequently modified. This option cannot be introduced with deterministic actions with a varying MASSE.
- ☐ MASSE TEMPORELLE: The masses matrix is updated at the start of each time interval and remains unchanged during the search for equilibrium. This is the default option. The masses are the masses of the structure and the applied masses.
- ☐ MASSE EXACTE: The masses matrix is updated at the start of each time interval and during the search for equilibrium. The masses are the masses of the structure and the applied masses.
- ☐ AMORTISSEMENT IGNORE: The damping matrix is not taken into consideration. If this option is introduced, no AMORTISSEMENT STRUCTURAL must be specified. This option is prohibited in the presence of AMORTISSEMENTS DIRECTS.
- ☐ AMORTISSEMENT INITIAL: The damping matrix is calculated once and for all at the start of the calculation and is not subsequently modified. This option is prohibited in the presence of AMORTISSEMENTS DIRECTS.
- ☐ AMORTISSEMENT TEMPOREL: The damping matrix is updated at the start of each time interval and remains unchanged during the search for equilibrium. This is the default option.
- ☐ AMORTISSEMENT EXACT: The damping matrix is updated at the start of each time interval and during the search for equilibrium.

The MASSES matrix and the AMORTISSEMENTS matrix must respect the table of compatibilities shown below.

DAMPING ➔	**IGNORE**	**INITIAL**	**TEMPORAL**	**EXACT**
MASSE IGNOREE	yes	-	-	-
MASSE INITIALE	-	yes	-	-
MASSE TEMPORELLE	-	-	yes	-
MASSE EXACTE	-	-	-	yes
AMORTISSEMENT DIRECT	-	-	yes	yes
AMORTISSEMENT RAYLEIGH	-	yes	yes	yes
AMORTISSEMENT MODAL	-	yes	yes	yes

NEWMARK METHOD

The differential system is solved using the Newmark method. The parameters p_1 and p_2 that control the temporal discretization are defined as follows:

$$\dot{U}(t+\Delta t)=\dot{U}(t)+\Delta t\,((1-p_1)\ddot{U}(t)+p_1\ddot{U}(t+\Delta t))$$

$$U(t+\Delta t)=U(t)+\Delta t\,\dot{U}(t)+\frac{\Delta t^2}{2}((1-p_2)\ddot{U}(t)+p_2\ddot{U}(t+\Delta t))$$

Caution: the definition of coefficient p_2 differs according to the authors of a factor 2. Here, the factor of the term of acceleration equals $p_2/2$, while for other authors it equals p_2. The latter is given at a value twice as large as in the case of the other convention.

The NEWMARK method is unconditionally superstable if the following conditions exist:

$$p_1 \geq \frac{1}{2}$$

$$p_2 \geq \frac{1}{2}\left(p_1+\frac{1}{2}\right)^2$$

For $p_1=0.50$ and $p_2=0.50$, the NEWMARK method does not introduce any numerical damping. These are the values taken by default.

The NEWMARK method only introduces numerical damping if $p_1 > \frac{1}{2}$. In this case, convergence is optimal for $p_2 = \frac{1}{2}\left(p_1+\frac{1}{2}\right)^2$.

The numerical damping develops from a frequency equal to $\frac{1}{10\,\Delta t}$ and reaches its maximum from the frequency $\frac{1}{\Delta t}$. For example, by fixing $p_1 = 0.6$ et $p_2 = 0.6050$, we obtain numerical damping of the order of 3% for all frequencies higher than $\frac{1}{\Delta t}$.

By fixing $p_1 = 0.8$ et $p_2 = 0.8450$, we obtain numerical damping of the order of 7% for all frequencies higher than $\frac{1}{\Delta t}$.

Therefore, the term p_1 controls the numerical damping. The higher the term, the greater the numerical damping.

SEEDING

The seeding number is related to stochastic actions. It defines a reproducible temporal scenario, because it initializes the "random" function used to generate the random signals. For two successive calculations in two distinct sessions to be independent, the difference between the two seedings must be at least equal to 3*nnoe*n_f*nb_sEries, where nnoe is the number of nodes impacted by the wind. nb_sEries is the entire value of t_fin/To_r.

INDEXES

Indexes are used to specify a temporal range within the complete range. These indexes can be used under the following conditions. A first calculation established a temporal index that causes an extreme effect in a linear calculation. It may be interesting to simulate the behavior of the structure in a non-linear calculation for a temporal range that contains the extreme effect, but not completely, in order to shorten the calculation time. It is advisable to take a duration equal to several times the first period of vibration.

KINETICS

By default, the displacements of the structure are calculated in the ABSOLU reference frame, i.e., in the event of an ACTION DETERMINISTE of ACCELERATION or DEPLACEMENT type, in the non-displaced reference frame. With the RELATIVE option, they are calculated in the reference frame in movement created by the ACTIONS DETERMINISTES, i.e., in the local reference frame of the supports.

In the absence of an ACTION DETERMINISTE of the ACCELERATION or DEPLACEMENT type, this option has no effect.

The speeds and accelerations are always evaluated in the absolute reference frame.

REPORTS AND EXTREMES

The solicitation, stress, displacement, etc. reports corresponding to each step of the calculations in the Newmark method can be EDITES and/or ENREGISTRES in the database at a frequency specified by the user.

The extreme values taken by these reports throughout the temporal calculation can be EDITES and/or ENREGISTRES in the database.

NON-LINEAR OPTIONS

These options are used to control the non-linear calculation when the saved STRUCTURE is defined in this mode. The options are: SECTION, DEPLACEMENT, FORCE and PONDERATION of the CALCUL NON LINEAIRE command of the PH3 module.

TEMPORAL RANGE AND TIME INTERVAL

The temporal calculation range is set by the user using the DEBUT and FIN parameters. This temporal range defines the time basis for the application of the ACTIONS DETERMINISTES. The temporal discretization PAS is also set by the user. For the REPONSE TEMPORELLE to be meaningful, it is advisable to use a time interval corresponding to one 10th of the smallest period of the meaningful mode of vibration in the structure.

Conditions of use

- This command must be preceded by the ACTION and STRUCTURE commands.
- The identification number of an EFFET already recorded by the PH3, ENV, DYN or ETU modules (according to the terminology used by the ETU module, see Table 12.1) cannot be used to designate a recorded modal REPONSE.
- If AMORTISSEMENTS are specified explicitly or by default, they must be defined beforehand.
- If the AMORTISSEMENTS refer to MODES, then the MODES must be defined beforehand.
- Several REPONSE TEMPORELLE commands may succeed one another, unless an ACTION ACCIDENTELLE is specified. No other REPONSE TEMPORELLE can follow the response that processed the accidental action.

Methodological advice

- This command can be used concurrently with the TEMPORELLE method by SUPERPOSITION MODALE in the case of dynamic calculations that are linear or linearized by the RIGIDITE TANGENTE option and the truncation of high-frequency modes.
- It is advisable to perform a static linear calculation before starting a dynamic linear calculation that must, itself, precede the non-linear dynamic calculation. This serves to validate the calculation data at each step.
- The calculations of the MASSE and AMORTISSEMENT matrices that are updated at the start of each time interval are usually sufficient, except in the presence of an AMORTISSEMENT DIRECT or a CALCUL NON LINEAIRE, which may require EXACTS calculations.
- The time interval must be chosen with care. A time interval of about 1/10 of the shortest significant period of the structure being studied is used.

This shortest period can be estimated by considering the shortest physical elements of the structure and we will take it for a bending bar:

$$\frac{1}{T_{phy}} = \frac{1}{2\pi}\max\left(2\frac{c}{L}, 2\sqrt{\frac{129I}{A}}\frac{c}{L^2}\right) \quad avec \quad c = \sqrt{\frac{\rho}{E}}$$

$$\Delta t_{phy} = \frac{T_{phy}}{10}$$

in which E is the modulus of elasticity, ρ is the density, L is the length, A is the surface area and I is the inertia.

- In the event of a stochastic action of the wind type, t_debut must equal 0 and t_fin must be a multiple of To_r for the statistics on the spectral densities to be meaningful.
- For vehicle movements, the time interval is calculated as a function of the speed of the vehicle and the length of the movement on the structure for a given time interval. This length is a sub-multiple of the length of the shortest longitudinal bar on which the vehicle moves. If L is the length of the element and V the speed of the vehicle, then:

$$\Delta t_{cin} = \frac{1}{10}\frac{L}{V}$$

$$\Delta t = \min\left(\Delta t_{cin}, \Delta t_{phy}\right)$$

- The conditions of the damping must be closely studied. The matrix of the RAYLEIGH damping alone underestimates the damping between the two frequencies chosen for calibration purposes, but the calculations are faster. The modal damping is more precise, provided that all the modes to be damped are pre-calculated, but the calculation times are longer. In a first analysis, a calculation can be made with only a RAYLEIGH damping. If this option is too severe, then the modal damping can be used.
- If the high-frequency modes are to be filtered, then the CALCUL MODAL FILTRE option is used, which eliminates the effect of the frequency modes higher than f_c.
- The introduction of parasite modes of numerical origin into the temporal calculation can be detected by comparing a linear temporal calculation and a linear temporal calculation by modal superimposition. If the temporal calculation shows interferences, then the parasite modes must be filtered with the CALCUL MODAL FILTRE option.
- If the numerical interference continues, then it is possible to also introduce numerical damping for a given cut frequency; fc. In this case, a time interval value is taken of the order of:

$$\Delta t_c = \frac{1}{f_c} \quad \Delta t_{num} = \frac{\Delta t_c}{10}$$

$$\Delta t = \min\left(\Delta t_{cin}, \Delta t_{phy}, \Delta t_{num}\right)$$

since we have already seen that the NEWMARK numerical damping starts at a frequency of the order of $\frac{1}{10\Delta t_c}$.

We can check that the numerical damping does not interfere with the low frequencies by comparing with a temporal calculation by modal superimposition as part of a linear calculation.

Examples

```
$ ---------------------------------------------------------------------------
$ classic temporal response with rayleigh damping
$ the calculation starts at 0 seconds and ends at 1 second
$ the graduation of the time is 1/100 of a second
$ the editions and recordings take place at a rate of 2
$ the extremas are edited and saved
$ ---------------------------------------------------------------------------
REPONSE  TEMPORELLE
    'EXEMPLE'
    AMORTISSEMENT STRUCTURAL RAYLEIGH 0.10 0.20
    DEBUT 0 FIN 1 PAS 0.01
    ETAT EDITER ENREGISTRER 1 PAS 2
    EXTREMAS EDITER ENREGISTRER 200;

$ ---------------------------------------------------------------------------
$ classic temporal response with rayleigh damping calculated
$ on the basis of a frequency of 1 and 100 Hertz
$ the calculation starts at 0 seconds and ends at 2 seconds.
$ the graduation of the time is 1/100 of a second
$ the editions and recordings of the movement take place at a rate of 2
$ the extremas of displacement are edited and saved
$ ---------------------------------------------------------------------------
REPONSE  TEMPORELLE
    'EXEMPLE'
    AMORTISSEMENT STRUCTURAL RAYLEIGH CALCULE 1 100
    DEBUT 0 FIN 2 PAS 0.01
    ETAT EDITER DEPLACEMENTS ENREGISTRER 1 PAS 2
    EXTREMAS EDITER ENREGISTRER 300 DEPLACEMENTS;

$ ---------------------------------------------------------------------------
$ classic temporal response with rayleigh damping calculated
$ element by element on the basis of the 5th harmonic
$ all the matrices are calculated exactly
$ the calculation starts at 0 seconds and ends at 2 seconds
$ the graduation of the time is 1/100 of a second
$ the editions and recordings of the normal stresses take place
$ at a rate of 2
$ the extremas of the normal stresses are edited and saved
$ ---------------------------------------------------------------------------
REPONSE  TEMPORELLE
    'EXEMPLE'
    AMORTISSEMENT STRUCTURAL RAYLEIGH CALCULE HARMONIQUE 5
    MATRICE AMORTISSEMENT EXACTE
    MATRICE MASSE EXACTE
    DEBUT 0 FIN 2 PAS 0.01
    ETAT EDITER CONTRAINTES NORMALES ENREGISTRER 1 PAS 2
    EXTREMAS EDITER ENREGISTRER 200 CONTRAINTES NORMALES;

$ ---------------------------------------------------------------------------
$ classic temporal response with classic modal damping
$ the matrix of rigidity is the matrix of tangential rigidity
$ the calculation starts at 0 seconds and ends at 2 seconds
$ the graduation of the time is 1/100 of a second
$ the extremas are edited and saved
$ ---------------------------------------------------------------------------
REPONSE  TEMPORELLE
    'EXEMPLE'
    AMORTISSEMENT STRUCTURAL MODAL 0 5.
    MATRICE RIGIDITE TANGENTE
    DEBUT 0 FIN 2 PAS 0.01
    EXTREMAS EDITER ENREGISTRER 200;

$ ---------------------------------------------------------------------------
$ classic temporal response with calculated modal damping
$ the matrix of rigidity is the matrix of tangential rigidity
$ the calculation starts at 0 seconds and ends at 2 seconds
$ the graduation of the time is 1/100 of a second
$ the extremas are edited and saved
```

```
$ a tack is specified for a temporal wind calculation
$ the sequence between indices 80 and 120 is replayed
$ ---------------------------------------------------------------------------
REPONSE  TEMPORELLE
    'EXEMPLE'
    AMORTISSEMENT STRUCTURAL MODAL CALCULE  0 5..
    MATRICE RIGIDITE TANGENTE
    DEBUT 0 FIN 2 PAS 0.01
    SEMENCE 1
    INDICES    80 120
    EXTREMAS EDITER ENREGISTRER 200;

$ ---------------------------------------------------------------------------
$ static temporal response: the dynamic effects are ignored
$ the calculation starts at 0 seconds and ends at 2 seconds
$ the graduation of the time is 1/100 of a second
$ the extremas are edited and saved
$ ---------------------------------------------------------------------------
REPONSE TEMPORELLE
    'EXEMPLE'
    MATRICE MASSE IGNOREE
    MATRICE AMORTISSEMENT IGNOREE
    DEBUT 0 FIN 2 PAS 0.001
    EXTREMAS EDITER ENREGISTRER 200;
```

Related commands

DYNAMIQUE ; MASSES ; MODES ; SELECTION MODES ;

AMORTISSEMENT DIRECT ; AMORTISSEMENT STRUCTURAL ;

AMORTISSEMENT ELEMENTAIRE ; ACTION DETERMINISTE ;

ACTION ACCIDENTELLE; ACTION STOCHASTIQUE VENT

11.22 - TEMPORAL RESPONSE BY SUPERIMPOSITION

REPONSE TEMPORELLE SUPERPOSITION_MODALE titre_reponse

$$\text{AMORTISSEMENT STRUCTURAL}\left\{\begin{array}{ll}\text{MODAL} & \\ \text{MODAL} & \text{CALCULE}\end{array}\right\}$$

$$[\text{SEMENCE} \quad \text{no\_semence}]$$

$$[\text{CINEMATIQUE} \quad \left\{\begin{array}{l}\underline{\text{ABSOLUE}}\\ \text{RELATIVE}\end{array}\right\}]$$

$$\overset{*}{[}\left\{\begin{array}{llll}\text{ETAT} & [\text{EDITER}\,[\text{options\_editions}]] & [\text{ENREGISTRER no\_etats}] & \text{PAS pas\_e}\\ \text{EXTREMAS} & [\text{EDITER}] & [\text{ENREGISTRER no\_reponse}] & [\text{options\_extremas}]\end{array}\right\}]$$

$$\text{DEBUT t\_debut} \quad \text{FIN} \quad \text{t\_fin} \quad \text{PAS} \quad \text{t\_pas};$$

$$[\text{options\_editions}] = [\text{options\_extremas}] = [\;{}^{*}\left\{\begin{array}{ll}\text{ACCELERATIONS} & \\ \text{VITESSES} & \\ \text{DEPLACEMENTS} & \\ \text{REACTIONS} & \\ \text{EFFORTS} & \text{ELEMENTS}\\ \text{EFFORTS} & \text{SECTIONS}\\ \text{CONTRAINTES NORMALES} & \\ \text{CONTRAINTES TANGENTES} & \end{array}\right\}]$$

Parameters

- titre_reponse: title given to the temporal response (character string).
- no_semence: number of the stochastic seeding to be considered for a stochastic simulation (1 by default).
- no_etats: record number of the various successive temporal reports, positive and less than 999. This number is independent of the effects numbers.
- no_reponse: record number of any positive responses, positive and less than 90_000. This number must be compatible with the effects numbers already recorded.
- pas_e: frequency of creation or recording of the successive reports. Only report numbers that are multiples of this frequency are created and/or recorded. 0, pas_e, 2 pas_e, etc.;
- t_debut and t_fin: initial and final times of the temporal calculation in seconds, with $t\_debut \geq 0$ and $t\_fin < 9999$;
- t_pas: time increment in seconds to be applied between two instants of temporal calculation. This is the temporal discretization interval.

Functions

This command starts the calculation of the temporal response by modal superimposition to an action or several deterministic or stochastic actions. These actions must be defined in FORCE, ACCELERATION or DEPLACEMENT. The only mandatory parameters are the time in seconds of the start and end of the calculation of the temporal response and the temporal discretization interval. The considered structure is the structure defined by the STRUCTURE command. The calculations are made according to the MODALE basis defined by the user with the MODE and SELECTION MODES commands.

The damping taken into consideration is calculated using different methods, according to the AMORTISSEMENTS ELEMENTAIRES or MODAUX introduced by the user. The user must pay attention to the conditions of the damping, because they determine the structure's response. AMORTISSEMENTS DIRECTS are forbidden.

STRUCTURAL DAMPING

The following options apply to AMORTISSEMENTS STRUCTURALS:

- MODAL: The damping is fixed mode by mode using the AMORTISSEMENT STRUCTUREL MODAL command.
- MODAL CALCULE: The damping is calculated using the AMORTISSEMENTS ELEMENTAIRES introduced by the user.

SEEDING

The seeding number is related to stochastic actions. It defines a reproducible temporal scenario, because it initializes the "random" function used to generate the random signals. For two successive calculations in two distinct sessions to be independent, the difference between the two seedings must be at least equal to 3*nnoe*n_f*nb_sEries, where nnoe is the number of nodes impacted by the wind. nb_sEries is the entire value of t_fin/To_r.

KINETICS

By default, the displacements of the structure are calculated in the ABSOLU reference frame, i.e., in the event of an ACTION DETERMINISTE of ACCELERATION or DEPLACEMENT type, in the non-displaced reference frame. With the RELATIVE option, they are calculated in the reference frame in movement created by the ACTIONS DETERMINISTES, i.e., in the local reference frame of the supports.

In the absence of an ACTION DETERMINISTE of the ACCELERATION or DEPLACEMENT type, this option has no effect.

The speeds and accelerations are always evaluated in the absolute reference frame.

REPORTS AND EXTREMES

The solicitation, stress, displacement, etc. reports corresponding to each step of the calculations in the Newmark method can be EDITES and/or ENREGISTRES in the database at a frequency specified by the user.

The extreme values taken by these reports throughout the temporal calculation can be EDITES and/or ENREGISTRES in the database.

TEMPORAL RANGE AND TIME INTERVAL

The temporal calculation range is set by the user using the DEBUT and FIN parameters. This temporal range defines the time basis for the application of the ACTIONS DETERMINISTES. The temporal discretization PAS is also set by the user. For the REPONSE TEMPORELLE to be meaningful, it is advisable to use a time interval corresponding to one 10^{th} of the smallest period of the meaningful mode of vibration in the structure.

In the event of a stochastic action of the wind type, t_debut must equal 0 and t_fin must be a multiple of To_r for the statistics on the spectral densities to be meaningful.

Conditions of use

- This command must be preceded by the ACTION, STRUCTURE, MODES and AMORTISSEMENT commands.
- The identification number of an EFFET already recorded by the PH3, ENV, DYN or ETU modules (according to the terminology used by the ETU module, see Table 12.1) cannot be used to designate a recorded modal REPONSE.
- Several REPONSE TEMPORELLE SUPERPOSITION commands can succeed one another.
- This command must not be preceded by an ACTION ACCIDENTELLE.
- Only winds without any AMORTISSEMENT AERODYNAMIQUE are accepted.
- This command must not be preceded by any ACTION DETERMINISTE of the type POIDS or MASSE.
- This command must not be preceded by an AMORTISSEMENT DIRECT.

Methodological advice

- The conditions of the damping must be closely studied.
- The time interval must also be chosen with care.
- It is advisable to make a static linear calculation before making a dynamic linear calculation. This serves to validate the calculation data at each step.

Examples

```
$ ---------------------------------------------------------------------------
$ temporal response by modal superposition
$ the calculation begins at 0 seconds and ends at 1 second.
$ the graduation of the time is 1/100 of a second
$ the editions and recordings take place at a rate of 2
$ the extremas are edited and saved
$ ---------------------------------------------------------------------------
REPONSE  TEMPORELLE SUPERPOSITION_MODALE 'SUPERPOSITION MODALE'
    AMORTISSEMENT STRUCTURAL MODAL 0.0 0.0
    DEBUT 0 FIN 1 PAS 0.01
    ETAT EDITER ENREGISTRER 1 PAS 2
    EXTREMAS EDITER ENREGISTRER 200;
```

Related commands

DYNAMIQUE ; MASSES ; MODES ; SELECTION MODES ;

AMORTISSEMENT STRUCTURAL ; AMORTISSEMENT ELEMENTAIRE ;

ACTION DETERMINISTE ; ACTION STOCHASTIQUE VENT

11.23 - END

FIN;

Functions

This command causes the end of a session and stops the execution of the DYN module; any following commands are ignored.

Examples

```
DYNAMIQUE      $ CHECK
TITRE ’VIADUC D’’ACCES B, CALCUL AU VENT, SESSION 1’;
..............
FIN;
```

Chapter 12

Studies

Introduction

Appendix A provides the full wording of the document referenced in this chapter under the condensed name: "BPEL (99)".

This chapter also refers to certain conventions and notations of the reference manual of the CDS software (calculation of reinforced and prestressed to limit state concrete sections).

Functions of the ETU module

This module provides access to various "EFFECTS" recorded in a database by the PH3, ENV and DYN modules, and itself, for EDITING, or EXPORT (in a form that some spreadsheets can accept).

Their compatible results can be COMBINED, WEIGHTED or ENVELOPED (for some areas of STUDY recorded by the ENV module, and certain components), together, or with data acquired directly. The results of these calculations can be edited, exported, REGISTERED in the database, or intervene in other calculations in chain combinations, weights or envelopes.

In addition to this processing, admissible TANGENTIAL STRESSES and minimum amounts of PASSIVE STEEL to be implemented can be calculated for a particular purpose, according to different regulatory criteria.

Finally, for a state of the structure associated with a particular effect, the ETU module generates a series of commands from the CDS software containing all the data needed for prestressed section calculations of an area of study (apart from their passive steel).

Effects handled

The term "effect" is taken in its broadest sense and covers the following entities:

- effects of LOAD CONDITIONS, a STATE or a MODE of buckling, recorded by the PH3 module;
- envelope effects of an ACTION, recorded by the ENV module;
- effects of a vibration MODE, or modal RESPONSE to turbulent wind, recorded by the DYN module;
- EFFECT introduced directly in data of the ETU module;
- WEIGHT (favorable and unfavorable) of a COMBINATION EFFECT or ENVELOPE effect of several EFFECTS recorded by the ETU module in a previous session, or produced in the current session.

Operations on these general effects may concern the DISPLACEMENT of nodes, REACTIONS of supports, FORCES in element or section reference frames, or normal or tangential STRESSES, and are not limited.

Admissible tangential stresses and minimum amounts of passive steel are also considered special effects that we cannot associate with each other, or other types of effects.

General conditions

The use of this module is subject to the prior registration, in the database, of at least a base effect from the PH3, ENV or DYN modules, and at least an area of study by the ENV module.

The results recorded during a session will be taken up by the ETU module in a later session or viewed by the RES module.

Data analysis mode

The ETU module end delimiter commands can be written completely freely (from the point of view of their breakdown into lines), the standard labels integrated in their presentation are purely indicative.

They are analyzed in full, in the order of their introduction (interpretation); the processing of effects, however, operates by looping on the areas of study chosen, via command blocks delimited appropriately; the ETU module can be instructed to just check commands, without executing them.

Editing

Commands are echoed as their interpretation proceeds; any erroneous commands are followed by error messages.

The results file contains only the effects for which editing was requested; the calculations of minimum passive steel quantities produce intermediate results that are always edited.

File export

It contains all the commands generated for the CDS software or all effects for which an export request has been made.

Contents

12.1 - Studies

$$\text{ETUDES} \quad [\ {}^{*}\left\{ \begin{matrix} \text{VERIFIER} \\ \text{TITRE} \quad \text{titre\_session} \end{matrix} \right\}]\ ;$$

Parameters

- titre_session: title attributed to the command file that will be reproduced at the head of the first results of the ETU module, if provided (string).

Functions

This command identifies a results study file and starts a "session" using the ETU module.

In VÉRIFICATION mode, the ETU module checks the syntax and logic commands, without performing the calculations, editing, exports or requested recordings; the number of detectable errors is unlimited.

In EXÉCUTION mode (VERIFIER option not used), calculation commands, editing, exporting or recording, declared correct, are executed.

By default, the session title is the title of the main model (first TITRE command of the PH1 module).

Conditions of use

- Must be at the beginning of the command file.

Methodological advice

- Always check the commands before starting an important calculation.

Examples

```
ETUDES  TITRE  'COMBINAISONS REGLEMENTAIRES, ETAPE 1';
.......
.......
FIN;
```

This ETUDES command label over two lines enables or disables the VÉRIFICATION mode, removing or restoring the "$" character of the first line.

```
ETUDES  $ CHECK
;
.......
.......
FIN;
```

Related commands

RAPPELER ; COMBINAISON ; PONDERATION ; ENVELOPPE ; TAULIMITES

ACIERS ; CDS ; COMMENTAIRE ; EDITER ; EXPORTER ; ENREGISTRER ; FIN

12.2 - TITLE

TITRE titre_courant ;

Parameters

- titre_courant: current identification title that will be reproduced in a box, in the results of the ETU module, outside the command echo (string).

Functions

This command is to redefine the current title.

Conditions of use

- ☐ This optional command can be entered multiple times at various stages of the calculation (outside the blocks that define the TRAITEMENT and FINTRAITEMENT commands).

Examples

```
ETUDES;
...........
TITRE  'ETUDE DES REACTIONS D''APPUIS';
TRAITEMENT
...........
FINTRAITEMENT;
TITRE  'ETUDE DES EFFORTS EN REPERES ELEMENTS';
TRAITEMENT
...........
FINTRAITEMENT;
FIN;
```

Related commands

TRAITEMENT ; FINTRAITEMENT

12.3 - RECALL

RAPPELER [TERMINAL] ;

Functions

This command is to edit the list of effects produced by the PH3, ENV, DYN and ETU modules, present in the database, with their types, numbers and identification labels.

With the TERMINAL option, this list includes the command echo (on screen).

Without the TERMINAL option, it is inserted into the output file.

Conditions of use

- ❐ This optional command can be entered multiple times at various stages of the calculation (outside the blocks that define the TRAITEMENT and FINTRAITEMENT commands).

Examples

```
ETUDES;
$ reminder on screen of the effects present in the database
RAPPELER  TERMINAL;
FIN;
```

```
ETUDES;
...........
TRAITEMENT
...........
ENREGISTRER  ...
ENREGISTRER  ...
...........
FINTRAITEMENT;
$ the list of effects previously present in the database
$ and of the effects recorded by the ETU module during the session
$ will be inserted in the results file
RAPPELER;
FIN;
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.4 - PROCESSING

TRAITEMENT

$$\left\langle \left\{ \begin{array}{l} \text{ETUDE} \quad \text{no\_domaine} \\ \text{DEPLACEMENTS} \\ \text{REACTIONS} \quad \text{STRUCTURE} \quad \text{nom\_structure} \\ \text{EFFORTS} \quad \text{SECTIONS} \quad [\text{POUTRE} \quad \text{no\_poutre}] \\ \left\{ \begin{array}{l} \text{EFFORTS} \quad \text{ELEMENTS} \\ \text{CONTRAINTES} \quad \left\{ \begin{array}{l} \text{NORMALES} \\ \text{TANGENTES} \end{array} \right\} \end{array} \right\} \quad [\left\{ \begin{array}{l} \text{POUTRE} \quad \text{no\_poutre} \\ \text{SECTION} \quad \text{nom\_section} \\ \text{STANDARD} \end{array} \right\}] \\ \text{EFFORTS} \quad \text{ELEMENTS} \quad [\left\{ \begin{array}{l} \text{BIARTICULATIONS} \\ \text{ARTICULATIONS} \end{array} \right\}] \end{array} \right\} \right.$$

$$\left. [\left\langle \text{COMPOSANTE} \quad \text{no\_cpp} \quad [\text{CONCOMITANTES} \quad \langle \text{no\_cpc} \rangle_{\text{nb\_cpc}}] \right\rangle_{\text{nb\_cpp}}] \right\rangle_{\text{nb\_domaines}} \quad ;$$

| ....

| commandes de traitement

| ....

| FINTRAITEMENT;

Parameters

The number of numbered (explicit) or unnumbered (implicit) areas of study, nb_domaines, is implicitly defined by the number of calls to the ETUDE option or to its concurrent options at the same level.

The numbers of principal components and concomitant components to a principal component (respectively nb_cpp and nb_cpc) are implicitly defined by the numbers of corresponding values provided.

- no_domaine: number of an area of study for which the following processing will be performed, defined by an ETUDE command of the ENV module (see Chapter 9); if the ETUDE option is not used, it applies to an unnumbered area;
- nom_structure: name of the saved structure for which the support reactions are being processed;
- no_poutre: the beam number for which FORCES are processed in SECTION or ELEMENT reference frames, or the STRESSES;
- nom_section: name of a standard section, defined by a SECTION TYPE command of the PH1 module (see Chapter 6); all elements having received this standard section assignment are implicitly designated for the processing of EFFORTS in ÉLÉMENTS or CONTRAINTES reference frames;
- no_cpp, no_cpc: number of a principal study component, and an associated concomitant component, to be chosen according to the conventions of Table 1.1.

By default, all the principal components of the no_domaine area of study are studied and no concomitant components are studied.

All processing commands that follow, until the next FINTRAITEMENT command, apply successively to each area of study chosen, by *looping;* they may be a composition of any number of commands:

- LECTURE ;
- COMBINAISON, PONDERATION, ENVELOPPE ;
- TAULIMITES, ACIERS ;
- CDS;
- COMMENTAIRE ;
- EDITER, EXPORTER, ENREGISTRER.

Functions

This command starts a group of processing for specific areas of explicit or implicit study, some principal components, and some possible concomitant components (referred to simply as "processing").

With the DEPLACEMENTS option, displacements of all the nodes are processed.

With the REACTIONS option, all the support reactions are processed.

With the STANDARD, BIARTICULATIONS or ARTICULATIONS options, all standard elements (current not bi-articulated), bi-articulations or articulations are considered.

In the absence of further additional options, the following are considered: all beam elements for the study of forces in section reference frames, all elements for the study of forces in element reference frames, or all beam (and standard) elements for the study of stresses.

The related processing commands apply to the effects recorded in the database, acquired directly and/or generated during processing.

Besides its title for identification of the content, each effect is identified, for its management, with a keyword indicating its type, a unique identification number, and the name of an optionally attached saved structure (by the PH3 module).

The type of an effect is the operation which is the origin of its creation.

EFFECT type (keyword)	**Content**	**Is it attached to a saved structure?**
CHARGE	Effects of load conditions, acquired from data and recorded by the PH3 module Effects of a nodal load from an average wind, generated by the DYN module, re-read and recorded by the PH3 module	Sometimes
ETAT	Probable state, extremal, weighted or prestressed, recorded by the PH3 module	Sometimes
MODE	Effects of a buckling mode, recorded by the PH3 module	Sometimes
ACTION	Envelope effects of an action, recorded by the ENV module	Always
MODE	Effects of a vibration mode, recorded by the DYN module	Always
REPONSE	Effects of modal response to turbulent wind or an earthquake, recorded by the DYN module	Always
LECTURE	Envelope effects data acquired by the ETU module	Never
COMBINAISON	Combination of several effects, created by the ETU module	Never
PONDERATION	Favorable and unfavorable weighting of an effect, created by the ETU module	Never
ENVELOPPE	Envelope for several effects, created by the ETU module	Never
TAULIMITES	Allowable tangential stresses, created by the ETU module	Never
ACIERS	Minimum amounts of passive steel, created by the ETU module	Never

Table 12.1 – Types of effects

All effects are considered as envelope curves, with in each point, a maximum and a minimum value; for some "single" effects (LOAD CONDITIONS, probable or prestressed STATES, buckling or vibration MODES), these values are made equal by convention.

The commands that generate TAULIMITES or ACIERS type effects apply only to studies of CONTRAINTES TANGENTES, on the two components: TANGENTES and NORMALES; these commands are ignored when applied to other types of studies.

TAULIMITES or ACIERS type effects are called "terminal", because they cannot be the subject of a COMBINAISON, PONDÉRATION or ENVELOPPE.

Any effect created during processing and not registered in the database is kept in "memory" and can be invoked, as long as the next FINTRAITEMENT command is not reached.

After processing, it is deleted, and cannot be invoked in subsequent processing.

Only an effect recorded in the database during processing may be invoked in subsequent processing, on the same studies, in the current session or in a later session.

In the descriptors of commands where they are used, the effects are identified using a single syntax:

type_effet no_effet [STRUCTURE nom_structure]

where:

- type_effet: effect type;
- no_effet: identification number;
- nom_structure: name of the attached saved structure.

Conditions of use

- The number of areas of study used in the same processing action is not limited.
- All effects invoked during processing must exist in memory or in the database.
- The study component numbers chosen for each area must be compatible with each other and the domain type.

Methodological advice

- It is usual to reuse areas and components of study used in the ENVELOPPES commands of the ENV module (see Chapter 9).

Examples

In the example below, the first area of study, unnumbered, is a "support reaction", the area of study 21 is a "normal stress", and the area of study 31 is "forces in section reference frames".

The processing applied to them concerns:

- all components of the reactions of supports, with no concomitant components;
- only the component of the normal stresses;
- all components of the forces in the section reference frames, with all their concomitant components.

```
TRAITEMENT
$ by default, all the components are studied
$ without concomitant components
REACTIONS  STRUCTURE  STRUC_1
ETUDE  21
$ it is necessary to give the details in order to access the concomitant
$ components
  ETUDE  31
  COMPOSANTE  1  CONCOMITANTES  2    3    4    5    6
  COMPOSANTE  2  CONCOMITANTES  1    3    4    5    6
  COMPOSANTE  3  CONCOMITANTES  1    2    4    5    6
  COMPOSANTE  4  CONCOMITANTES  1    2    3    5    6
  COMPOSANTE  5  CONCOMITANTES  1    2    3    4    6
  COMPOSANTE  6  CONCOMITANTES  1    2    3    4    5;
  $ processing commands
  ..............
FINTRAITEMENT;
```

Exemple équivalent :

```
TRAITEMENT
  REACTIONS  STRUCTURE  STRUC_1
  COMPOSANTE  1  COMPOSANTE  2  COMPOSANTE  3
  COMPOSANTE  4  COMPOSANTE  5  COMPOSANTE  6
  ETUDE  21
  COMPOSANTE  1
  ETUDE  31
  COMPOSANTE  1  CONCOMITANTES  2    3    4    5    6
  COMPOSANTE  2  CONCOMITANTES  1    3    4    5    6
  COMPOSANTE  3  CONCOMITANTES  1    2    4    5    6
  COMPOSANTE  4  CONCOMITANTES  1    2    3    5    6
  COMPOSANTE  5  CONCOMITANTES  1    2    3    4    6
  COMPOSANTE  6  CONCOMITANTES  1    2    3    4    5;
  $ processing commands
  ..............
FINTRAITEMENT;
```

Exemple équivalent :

```
TRAITEMENT
REACTIONS  STRUCTURE  STRUC_1;
$ processing commands
  ..............
FINTRAITEMENT;
TRAITEMENT
  ETUDE  21;
  $ processing commands
  ..............
FINTRAITEMENT;
TRAITEMENT
  ETUDE  31
  COMPOSANTE  1  CONCOMITANTES  2    3    4    5    6
  COMPOSANTE  2  CONCOMITANTES  1    3    4    5    6
  COMPOSANTE  3  CONCOMITANTES  1    2    4    5    6
  COMPOSANTE  4  CONCOMITANTES  1    2    3    5    6
  COMPOSANTE  5  CONCOMITANTES  1    2    3    4    6
  COMPOSANTE  6  CONCOMITANTES  1    2    3    4    5;
  $ processing commands
  ..............
FINTRAITEMENT;
```

Related commands

TITRE ; RAPPELER ; LECTURE ; COMBINAISON ; PONDERATION ; ENVELOPPE

TAULIMITES ; ACIERS ; CDS ; COMMENTAIRE ; EDITER ; EXPORTER

ENREGISTRER ; FINTRAITEMENT

12.5 - READING

LECTURE no_effet titre_effet

$$\text{VALEURS} \quad \left\{ \begin{array}{l} \text{SIMPLES} \quad \left\langle \langle \text{v} \rangle_{\text{nb\_composantes}} \right\rangle_{\text{nb\_points}} \\ \text{ENVELOPPES} \quad \left\langle \langle \text{v\_max} \rangle_{\text{nb\_composantes}} \quad \langle \text{v\_min} \rangle_{\text{nb\_composantes}} \right\rangle_{\text{nb\_points}} \end{array} \right\} ;$$

Parameters

The numbers of components (a principal and any concomitants) and study points (respectively nb_composantes and nb_points) are implicitly set by the last TRAITEMENT command.

- no_effet: identification number of the effect to be created, positive and less than 90,000;
- titre_effet: title that will be remembered at each of its uses, in the results of the ETU module, menus and drawings of the RES module;
- v, v_max, v_min: value of the single effect, maximum and minimum values of the effect envelope, for a component and a point of the area of study under processing.

Functions

This command is to directly read a single effect (SIMPLES option) or envelope (ENVELOPPES option); the values of the single effect will be automatically split, identically.

The effect read is managed in the same way as an effect created by the ETU module (see TRAITEMENT command).

Conditions of use

- This optional command can be introduced several times during processing.
- The last TRAITEMENT command should only invoke a single area of study, for which only one principal component (and any concomitant components) are studied.
- A LECTURE type effect existing in memory is replaced, if it is reread with the same identification number.
- A LECTURE type effect existing in the database is not replaced unless an ENREGISTRER command is applied to an effect of the same type reread and with the same ID number.
- The identification number assigned to a LECTURE type effect is not to be used to designate an effect of a different origin (see Table 12.1).

Examples

Consider an area of study with five points, in which a principal component is studied, and two concomitant components (input values are fictitious).

```
TRAITEMENT  ETUDE  11  COMPOSANTE  1
                       CONCOMITANTES  2 3;
..............
$ cp: main component, cc: concomitant component
$ pe: point of study
LECTURE  101  'GRADIENT THERMIQUE'
VALEURS  SIMPLES
$   v(cp 1)   v(cc 2)   v(cc 3)
     100.0      50.0     300.0  $ pe 1
     200.0      25.0     350.0  $ pe 2
     350.0      65.0     600.0  $ pe 3
     425.0      80.0     505.0  $ pe 4
     600.0      25.0     250.0; $ pe 5
LECTURE  102  'CHARGES DE CHANTIER'
VALEURS  ENVELOPPES
$ v_max(cp 1) v_max(cc 2) v_max(cc 3) v_min(cp 1) v_min(cc 2) v_min(cc 3)
     100          30          250         -200          47          30 $ pe 1
     200          25          330         -120          30          50 $ pe 2
     225          37          450         -100          22          75 $ pe 3
     347          41          405          -57          12          47 $ pe 4
     720          63          309          -78         -20          79;$ pe 5
$ calculate, create, export and save commands
$ that may apply effects 101 and 102
..............
$ these commands authorize the future use of the effects read in
$ other processes in the current session, or in other sessions
ENREGISTRER  101;
ENREGISTRER  102;
..............
FINTRAITEMENT;
..............
```

Related commands

TRAITEMENT ; FINTRAITEMENT

12.6 - COMBINATION

COMBINAISON [QUADRATIQUE] no_combinaison titre_combinaison

$$\left\langle \begin{matrix} \left.\begin{cases} \text{FAVORABLE } k_{\text{fav}} \text{ DEFAVORABLE } k_{\text{def}} \\ \text{COEFFICIENT } \quad k \end{cases}\right\} \text{type\_effet no\_effet} \\ \text{[STRUCTURE nom\_structure]} \end{matrix} \right\rangle_{\text{nb\_effets}} \quad ;$$

Parameters

The number of effects to be combined, nb_effets, is implicitly defined by the data provided.

- no_combinaison: identification number of the combination to be created, positive and less than 90,000;
- titre_combinaison: title that will be remembered at each of its uses, in the results of the ETU module, the in menus and drawings of the RES module. This title is a character string delimited by single quotation marks (‘);
- k: weighting coefficient of the effect no_effet (negative, zero or positive value);
- k_{fav}, k_{def}: weight coefficients to be applied to maximum and minimum values of the effect no_effet, whether favorable or unfavorable. The expected value of k_{fav} is in the range [0.0; 1.0], and the expected value of k_{def} is greater than 1.0. If it is not the case, a warning message will appear without interrupting the calculation, which is done with the values entered by the user;
- type_effet, nom_effet, nom_structure: parameters identifying an effect to be combined (see TRAITEMENT command).

Functions

This command causes the calculation of an effect by linear (the default option), or quadratic (QUADRATIQUE option) combination, of several compatible effects, weighted individually.

The maximum and minimum values of the effect to be created (v_max_e and v_min_e) are calculated at each point and for each component of study according to the formulas below.

For a linear combination:

$$v\_max_e = \sum_{i=1,nb\_effets} \max(k_i \cdot v\_max_i,\ k_i \cdot v\_min_i) \tag{12.1}$$

$$v\_min_e = \sum_{i=1,nb\_effets} \min(k_i \cdot v\_max_i,\ k_i \cdot v\_min_i) \tag{12.2}$$

For a quadratic combination:

$$v\_max_e = +\sqrt{\sum_{i=1,nb\_effets} (k_i \cdot v\_max_i)^2} \tag{12.3}$$

$$v\_min_e = -\sqrt{\sum_{i=1,nb\_effets} (k_i \cdot v\_min_i)^2} \tag{12.4}$$

The created combination is managed under the conditions set in the "Functions" section of the TRAITEMENT command.

Conditions of use

- This optional command can be introduced several times during processing.
- All types of effects can be combined, except for the terminal types (see TRAITEMENT command).
- The effects to be combined must exist in the memory, or in the database.
- A COMBINAISON type effect existing in memory is replaced if it is reread with the same identification number.
- A COMBINAISON type effect existing in the database is not replaced if an ENREGISTRER command is applied to an effect of the same type and recalculated with the same ID number.
- The identification number assigned to a COMBINAISON type effect is not to be used also to designate an effect of different origin (see Table 12.1).
- The user must ensure that the syntax of the parameter titre_combinaison is correct: the oversight of a single quotation mark (‘) at the end induces a systematic error.

Methodological advice

- This command makes a linear combination of several effects. If the user wants to take into account the non-linear nature of the structure, he could use the phase module PH3 that makes non-linear calculations, by defining equivalent loads of the combination. Here are the main steps :
  - In the phase file, after having introduced the permanent loads, save the structure with the SAUVER command
  - Create then as many phase files as combinations of loads to study: files with the PHASES SUITE command allows the user to go on a phasing from a saved structure.
  - In each PHASE SUITE file, enter the appropriate loading cases thanks to the CAS DE CHARGE command, then use the ETAT command to record the results in the data base.

Examples

```
$ linear combination of the effects of the permanent loads, elevated
$ overloads and thermal gradients. All the combined effects are
$ attached to the same saved structure: STRUC1
COMBINAISON  1000  'C.P. + 1.10*A(L) + GRADIENT 5 DEGRES'
  FAVORABLE  1.00  ETAT      0  STRUCTURE  STRUC1
  FAVORABLE  1.10  ACTION  100  STRUCTURE  STRUC1
  FAVORABLE  1.00  CHARGE   11  STRUCTURE  STRUC1;

$ linear combination simulating the overall redistribution of the loads
$ by creep. No combined states are attached to a saved
$ stucture
$ STATE 1: state of the structure when brought into service
$ STATE 2: state of the supposed structure cast immediately on an arch
COMBINAISON  1001  '0.50*(S1+S2)'
  FAVORABLE  0.50  ETAT  1
  DEFAVORABLE  0.50  ETAT  2;

$ study of loads by section coordinates
$ the QUADRATIQUE option applies to the effects of turbulent wind
$ by default, the other combinations are formed in linear mode
COMBINAISON  QUADRATIQUE  3008
'VENT TURBULENT QUADRATIQUE EST'
  COEFFICIENT  3.50  REPONSE  800  STRUCTURE  FLEAU_2;
```

```
..............
COMBINAISON  3012
'ELS RARE : CHARGES PERMANENTES + PRECONTRAINTE MOYENNE + VENT QUADRATIQUE'
  COEFFICIENT  1.00  COMBINAISON 3008
  COEFFICIENT  1.00  CHARGE  850  STRUCTURE  FLEAU_2
  COEFFICIENT  1.00  ETAT    0    STRUCTURE  FLEAU_2;

COMBINAISON  100
'(0.80 OU 1.20)*SUPERSTRUCTURES + GRADIENT THERMIQUE'
  FAVORABLE  0.80  DEFAVORABLE  1.20  CHARGE 25
  COEFFICIENT  1. CHARGE 20 STRUC STRUserv ;
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.7 - WEIGHTING

PONDERATION no_ponderation titre_ponderation

FAVORABLE k_{fav} DEFAVORABLE k_{def}

type_effet no_effet [STRUCTURE nom_structure] ;

Parameters

- no_ponderation: identification number of the weighting to be creates, positive and less than 90,000;
- titre_ponderation: title that will be remembered at each of its uses, in the results of the ETU module, and in the menus and drawings of the RES module;
- k_{fav}, k_{def}: weight coefficients to be applied to maximum and minimum values of the effect no_effet, whether favorable or unfavorable. The expected value of k_{fav} is in the range [0.0; 1.0], and the expected value of k_{def} is greater than 1.0. If it is not the case, a warning message will appear without interrupting the calculation, which is done with the values entered by the user;
- type_effet, nom_effet, nom_structure: parameters identifying the effect to be weighted (see TRAITEMENT command).

Functions

This command causes the calculation of an effect by favorable and unfavorable weighting of another effect.

At each point and for each component of the study, its *maximum* value is weighted by k_{fav} when it is *negative,* and its *minimum* value is weighted by k_{def} when it is*positive*. According to the following formulas:

$$v\_max_e = \max(k_i \cdot v\_max_i,\ k_i \cdot v\_min_i) \tag{12.5}$$

$$v\_min_e = \min(k_i \cdot v\_max_i,\ k_i \cdot v\_min_i) \tag{12.6}$$

The created weighting is managed according to the conditions laid down in the "Functions" section of the TRAITEMENT command.

Conditions of use

- The conditions of use of the COMBINAISON command apply to the PONDÉRATION command (which uses only one effect).

Examples

```
$ weighting of the effects of a LOAD CASE due to superstructures
$ and attached to the saved structure: STRUC1
$ these effects are reduced if they are favorable (coefficient 0.80)
$ or increased if they are unfavorable(coefficient 1.20)
PONDERATION  100  ’(0.80 OU 1.20)*SUPERSTRUCTURES’
  FAVORABLE  0.80  DEFAVORABLE  1.20
  CHARGE  25  STRUCTURE  STRUC1;
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.8 - ENVELOPPE

ENVELOPPE no_enveloppe titre_enveloppe

$$\left\langle \left\{ \begin{array}{l} \text{FAVORABLE } k_{\text{fav}} \text{ DEFAVORABLE } k_{\text{def}} \\ \text{COEFFICIENT } \quad k \end{array} \right\} \text{type\_effet no\_effet} \quad [\text{STRUCTURE nom\_structure}] \right\rangle_{\text{nb\_effets}} \quad ;$$

Parameters

The number of items to be enveloped, nb_effets, is implicitly defined by the data provided.

- no_enveloppe: identification number of the envelope to be created, positive and less than 90,000;
- titre_enveloppe: title that will be remembered at each of its uses, in the results of the ETU module, and in the menus and drawings of the RES module;
- k: weighting coefficient of the effect no_effet (negative, zero or positive value);
- k_{fav}, k_{def}: weight coefficients to be applied to maximum and minimum values of the effect no_effet, whether favorable or unfavorable. The expected value of k_{fav} is in the range [0.0; 1.0], and the expected value of k_{def} is greater than 1.0. If it is not the case, a warning message will appear without interrupting the calculation, which is done with the values entered by the user;
- type_effet, nom_effet, nom_structure: parameters identifying an effect needing an envelope (see TRAITEMENT command).

Functions

This command causes the calculation of an effect per envelope of several compatible effects, weighted individually.

The maximum and minimum values of the effect to be created (v_max_e et v_min_e) are calculated at each point and for each component of study according to the formulas below:

$$v\_max_e = \max_{i=1,nb\_effets} (k_i \cdot v\_max_i,\ k_i \cdot v\_min_i) \tag{12.7}$$

$$v\_min_e = \min_{i=1,nb\_effets} (k_i \cdot v\_max_i,\ k_i \cdot v\_min_i) \tag{12.8}$$

The envelope created is managed according to the conditions laid down in the "Functions" section of the TRAITEMENT command.

Conditions of use

- The conditions of use of the COMBINAISON command apply to the ENVELOPPE command.

Examples

```
$ calculation of the enclosure effects of two combinations
ENVELOPPE  102;
'ENVELOPPE FINALE'
  COEFFICIENT  1.00  COMBINAISON  100
  COEFFICIENT  1.00  COMBINAISON  101;

$ structure with four supports
$ ------------------------------------
$ calculation of the enclosure effect of LOAD CASES simulating settlements
$ of supports that are exclusive of one another:
$ LOAD CASE 10, applied to the first support, causes settlement that is
$ 10.570 times lower than the settlement considered in the enclosure. The
$ same is true of LOAD CASES 11, 12 and 13 applied to supports 2, 3 and 4
$ that produce settlements 15.750, 19.101 and 17.103 times too weak
ENVELOPPE  100;
'ENVELOPPE DES TASSEMENTS D''APPUIS'
  COEFFICIENT  10.570  CHARGE  10
  COEFFICIENT  15.750  CHARGE  11
  COEFFICIENT  19.101  CHARGE  12
  COEFFICIENT  17.103  CHARGE  13;
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.9 - TAULIMITES

$$\text{TAULIMITES}\quad \text{no\_taulimites}\quad \text{titre\_taulimites}\quad \begin{Bmatrix}\textbf{CHALOS}\\ \textbf{BPELELS}\\ \textbf{BPELELU}\\ \textbf{BPHPELU}\\ \textbf{EC2ELS}\\ \textbf{EC2ELU}\\ \textbf{EC2ELUS}\ \beta_{\text{u}}\end{Bmatrix}$$

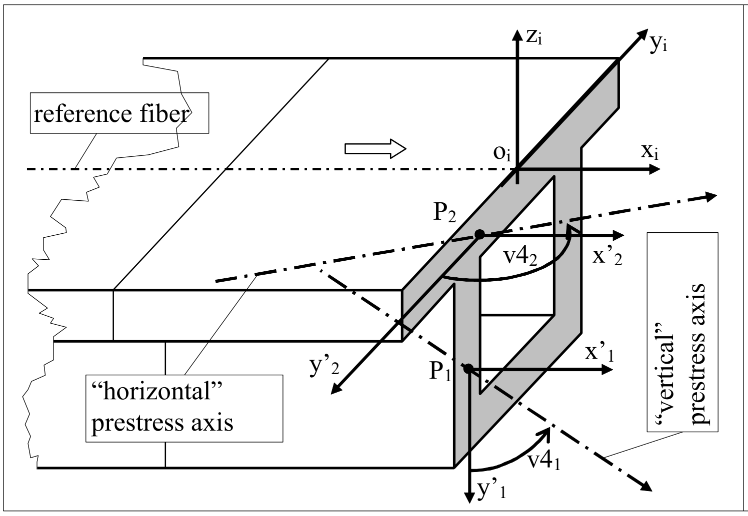
$$\text{RESISTANCE}\quad +\begin{Bmatrix}\text{TRACTION}\quad \text{v1}\\ \text{COMPRESSIO N}\quad \text{v2}\end{Bmatrix}\quad \text{COEFFICIENT}\quad \text{k}$$

$$[\text{ACIERS}\quad \begin{Bmatrix}\text{CONSTANTES}\quad \text{v3}\quad \text{v4}\\ \text{VARIABLES}\quad \text{nb\_points}\quad \langle \text{v3}\rangle_{\text{nb\_points}}\quad \langle \text{v4}\rangle_{\text{nb\_points}}\end{Bmatrix}]$$

$$\text{type\_effet}\quad \text{no\_effet}\quad [\text{STRUCTURE}\quad \text{nom\_structure}]\ ;$$

Parameters

- no_taulimites: identification number of a TAULIMITES type effect to be created, positive and less than 90,000;
- titre_taulimites: title that will be remembered at each of its uses, in the results of the ETU module, and in the menus and drawings of the RES module;
- v1, v2: strength of concrete in tension (in absolute value), and in compression ($\text{v2} > 4 \cdot \text{v1}$);
- k: reduction coefficient to be applied to the tangential limit stress, calculated according to the selected criteria;
- nb_points: total number of study points involved in the calculation;
- v3: normal stress induced in a study point, by the transverse prestress, on a side orthogonal to its axis;
- v4: angle (in degrees) between the transverse prestress axis, and that of the thin wall associated with a study point (which is in the plane of the generic section of the beam concerned); when this is a "through" prestress, its sense "follows" conventionally that of the reference fiber.
- β_u: angle (degrees) of the strut with respect to the mean fiber of the relevant beam from calculation; this value (between -90.0 and 90.0, limits excluded) is provided (with 1 EC2ELUS option) in accordance with Article EC2-1-1/AN 6.2.3(2). By default, this angle is calculated by the ETU module and bounded below at 30.0 degrees;



$o_ix_iy_iz_i$: generic reference section i

⇨ reference fiber traffic flow

$P_jx'_j$: axis parallel to o_ix_i passing by the calculation point j

$P_jy'_j$: axis reference thin wall at the the calculation point j

$v4_j$: angle between $P_jy'_j$ and the axis of the transverse prestress, counted positively from $P_jx'_j$ to $P_jy'_j$

Figure 12.1 - Transverse prestress of a beam cross section, conventions

With the CONSTANTES option, v3 and v4 are average values, for all study points involved in the calculation.

With the VARIABLES option, v3 and v4 are values provided for each of these study points, in order of definition.

- type_effet, nom_effet, nom_structure: parameters identifying the effect selected for calculation (see TRAITEMENT command).

Functions

This command is to calculate, for all points of an area of study of CONTRAINTES TANGENTES and NORMALES type and a given effect, admissible tangential stresses according to regulatory criteria to be chosen.

A TAULIMITES type effect is inferred from the effect studied, by replacing, in each point of the study, the real tangential stress (component 1), with the smallest difference, $\delta\tau$, existing between itself and the admissible minimum and maximum tangential stresses multiplied by the reduction coefficient k.

The value obtained is positive when the actual tangential stress is within the range of reduced admissible stresses; it is given by:

$$\delta\tau = \min(k \cdot \tau_{max} - \tau,\ \tau - k \cdot \tau_{min}) \qquad (12.9)$$

τ_{max} and τ_{min} are the calculated maximum and minimum admissible tangential stresses :

- according to the criterion of Chalos and Béteille (CHALOS option);
- according to BPEL, for a calculation at the service limit state (Article 7.2,2, BPELELS option), or a calculation at the ultimate limit state (Article 7.3,3, BPELELU option).
- According to Annex QQ of EC2.2 for ELS and Article 6.2.3 of EC2.2 for ELU

If the normal stress is outside the field of concrete strength, the component 1 of the TAULIMITES type effect is conventionally set at -1.0E +40.

The stresses used in the formulas below are calculated taking into account the transverse prestress.

CHALOS option

Values τ_{max} and τ_{min} are deducted from the following system of inequalities:

$$\tau\left[\tau-\sigma_\theta \sin(2\theta)\right] \le \sigma\cdot\sigma_\theta \cos^2\theta+\frac{\bar{\sigma}'}{\bar{\sigma}}(\bar{\sigma}'+\sigma+\sigma_\theta)\cdot(\bar{\sigma}-\sigma-\sigma_\theta) \tag{12.10}$$

$$\left[\tau-0.5\cdot\sigma_\theta \sin(2\theta)\right]^2 \le (\bar{\sigma}-\sigma_\theta \cos^2\theta)\cdot(\bar{\sigma}-\sigma-\sigma_\theta \sin^2\theta) \tag{12.11}$$

where:

- τ : tangential stress;
- θ : angle of orientation of the transverse prestress, with respect to the straight section;
- σ_θ : transverse normal stress;
- $\bar{\sigma}'$, $\bar{\sigma}$: strength of concrete in tension and compression (parameters v1 and v2);
- σ : longitudinal normal stress.

The current value of parameter v4 is used to determine the angle θ.

BPELELS option

Values τ_{max} and τ_{min} are deducted from the following system of inequalities:

$$\tau^2-\sigma_x\cdot\sigma_t \le 0.4f_{tj}\left[f_{tj}+\frac{2}{3}(\sigma_x+\sigma_t)\right] \tag{12.12}$$

$$\tau^2-\sigma_x\cdot\sigma_t \le 2\frac{f_{tj}}{f_{cj}}(0.6f_{cj}-\sigma_x-\sigma_t)\cdot\left[f_{tj}+\frac{2}{3}(\sigma_x+\sigma_t)\right] \tag{12.13}$$

where (according to BPEL):

- τ : tangential stress;
- σ_x: longitudinal normal stress;
- σ_t: transverse normal stress;
- f_{tj}, f_{cj}: concrete strength in tension and compression (parameters v1 and v2).

When σ_x is negative, the above inequalities are replaced by:

$$\tau^2 \le 0.4f_{tj}(f_{tj}+\frac{2}{3}\sigma_t) \tag{12.14}$$

The coefficient k’ of the Regulation corresponds to the parameter k.

BPELELU option

Values τ_{max} and τ_{min} are given by:

$$\tau_{\max}=\tau_{\min}=k\cdot f_{cj}\cdot\sin(2\beta_u) \tag{12.15}$$

The reduction factor (parameter k) is:

$$\frac{0.85}{3\gamma_b} \tag{12.16}$$

where (according to BPEL):

- f_{cj}: strength of concrete in compression (parameter v2);
- β_u: angle formed by the cracks with the mean fiber of the beam;

- γ_b: safety factor for concrete.

BPHPELU option

Values τ_{max} and τ_{min} are given by:

$$\tau_{\max} = \tau_{\min} = k \cdot f_{cj}^{2/3} \cdot \sin(2\beta_u) \tag{12.17}$$

The reduction factor (parameter k) is:

$$1.14 \cdot \frac{0.85}{3\ \gamma_b} \tag{12.18}$$

- These formulas are taken from Annex 14 of BPEL, dedicated to high performance concretes, f_{cj}, β_u and γ_b having the same meaning as above.

EC2ELS option

Values τ_{max} and τ_{min} are given by:

$$\tau_{\max} = \tau_{\min} = k\sqrt{\sigma_x.\sigma_y - \frac{5f_{ck} \times f_{ctk} \times (\sigma_x + \sigma_y + f_{ctk}) \times (4\sigma_x + 4\sigma_y - 5f_{ctk})}{(5f_{ck} + 4f_{ctk})^2}} \tag{12.19}$$

where, according to EC2-2 Expr (QQ.101):

- σ_x: longitudinal normal stress;
- σ_t: transverse normal stress;
- f_{ck}, f_{ctk}: concrete strength in tension and compression (parameters v1 and v2).
- *k:* weighting coefficient generally equal to 1;

EC2ELU option

Values τ_{max} and τ_{min} are given by:

$$\tau_{\max} = \tau_{\min} = k \cdot a_{cw} \cdot \nu_1 \cdot k \cdot f_{ck} \cdot \sin(2\beta_u) \tag{12.20}$$

where, according to EC2-2 Expr (6.14):

- a_{cw}: coefficient determined according to f_{ck};
- ν_1: coefficient determined according to fcd;
- f_{ck}: concrete strength in tension and compression (parameter v1).
- *k:* weighting coefficient such that:

$$f_{cd} = k.f_{ck} = \frac{\alpha_{cc}}{\gamma_c}.f_{ck}\ ;$$

EC2ELUS option

Same as above but with the value of β_u input.

Conditions of use

- This command applies to studies of CONTRAINTES TANGENTES and NORMALES types, on both components, whether the chosen principal component be the tangential stress or the normal stress; it is ignored if the concommitant component is not studied, or if applied to other types of studies.
- With the ACIERS and VARIABLES options, it normally applies to a single area of study; with the ACIERS and CONSTANTES options, it can be applied to several areas of study.
- The employment conditions of the COMBINAISON command also apply to the TAULIMITES command (which uses only one effect).

Examples

```
$ processing of three study domains of a beam (command block)
$ domain 10, of which the two components are alternately studied,
$ covers the CONTRAINTES TANGENTES and NORMALES, and certain sections
$ the TAULIMITES command is not executed for study domains
$ 1 and 11, which are of other types
$ The transverse prestress exists and its definition parameters are
$ assumed to be constant. In particular, the axes of the cables are
$ assumed to be parallel to those of the thin walls produced by the generic
$ sections
TRAITEMENT
  ETUDE  1
  ETUDE  10  COMPOSANTE  1  CONCOMITANTE  2
             COMPOSANTE  2  CONCOMITANTE  1
  ETUDE  11;
  ..............
  ENVELOPPE  1000 ...
  ..............
  TAULIMITES  100
  'CONTRAINTES TANGENTES ADMISSIBLES SOUS ENVELOPPE FINALE (ELS)'
  BPELELS
  RESISTANCE  TRACTION  300.0  COMPRESSION  3000.0
  COEFFICIENT  1.0
  ACIERS  CONSTANTES  200.0  0.0
  ENVELOPPE  1000;
  ..............
  EDITER  TAULIMITES  100;
  ENREGISTRER  TAULIMITES  100;
  ..............
FINTRAITEMENT;
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.10 - STEELS

$$\text{ACIERS}\quad \text{no\_aciers}\quad \text{titre\_aciers}\quad \left\{\begin{array}{l}\textbf{BPELELU}\\ \textbf{BPELELUS}\ \ \beta_{\text{u}}\\ \textbf{BPHPELU}\\ \textbf{BPHPELUS}\ \ \beta_{\text{u}}\\ \textbf{EC2ELU}\\ \textbf{EC2ELUS}\ \ \beta_{\text{u}}\end{array}\right\}$$

$$\text{RESISTANCE}\quad + \left\{\begin{array}{ll}\text{TRACTION} & \text{v1}\\ \text{COMPRESSION} & \text{v2}\end{array}\right\}\quad \text{COEFFICIENT}\quad \text{k}$$

$$[\text{ACIERS}\quad \left\{\begin{array}{l}\text{CONSTANTES}\quad \text{v3}\quad \text{v4}\\ \text{VARIABLES}\quad \text{nb\_points}\quad \langle \text{v3}\rangle_{\text{nb\_points}}\quad \langle \text{v4}\rangle_{\text{nb\_points}}\end{array}\right\}]$$

$$\text{type\_effet}\quad \text{no\_effet}\quad [\text{STRUCTURE}\quad \text{nom\_structure}]\ ;$$

Parameters

- no_aciers: identification number of an ACIERS type effect to be created, positive and less than 90,000;
- titre_aciers: title that will be remembered at each of its uses, in the results of the ETU module, and in the menus and drawings of the RES module;
- β_u: angle (in degrees) made by cracks with the mean fiber of the relevant beam from calculation; this value (between -90.0 and 90.0, limits excluded) is provided (with the BPELELUS, BPHPELUS or EC2ELUS options), regardless of the reality; in accordance with the provisions of Articles 7.3,23 and 7.3,24 of BPEL, and Article EC2-1-1/AN 6.2.3(2) of Eurocodes; with the other options BPELELU, BPHPELU or EC2ELU, this angle is calculated by the ETU module and bounded below at 30.0 degrees;
- v1, v2: strength of concrete in tension (in absolute value), and in compression ($\text{v2} > 4 \cdot \text{v1}$);
- k: coefficient of forces transferred to be balanced, per unit beam length, to the quantities of "transverse" steel bars to be positioned (see details in the "Functions" section);
- nb_points: total number of study points involved in the calculation;
- v3: normal stress induced in a study point, by the transverse prestress, on a side orthogonal to its axis; a zero value indicates no transverse prestress;
- v4: angle (in degrees) between the axis of the transverse steel bars (passive and possibly active), with that of the thin wall associated with a point of study (which is in the plane of the generic section of the beam concerned); when these are "through" bars, their sense "follows" conventionally that of the reference fiber (see Figure 12.1).

With the CONSTANTES option, v3 and v4 are average values, for all study points involved in the calculation.

With the VARIABLES option, v3 and v4 are values provided for each of these study points, in order of definition.

- type_effet, nom_effet, nom_structure: parameters identifying the effect selected for calculation (see TRAITEMENT command).

Functions

This command is to calculate, for all points of an area of study of CONTRAINTES TANGENTES and NORMALES type, and a given effect, the minimum transverse steel bar quantities required to meet a regulatory requirement.

An ACIERS type effect is inferred from the effect studied, by replacing, in each point of the study, the real tangential stress (component 1), by the calculated steel bar quantity.

Passive transverse steel bars (and possibly active) are assumed to have the same modifiable orientation, which is by default orthogonal to the mean fiber of the studied beam.

When the transverse prestress exists, the calculated steel bar quantities are assimilated to the total forces to be balanced, using passive and active steel bars.

The calculated steel bar quantity, *x,* is given by the following equation (involving parameters k and β_u, $\delta\tau_u$ of value 0 for the EC2ELU and EC2ELUS options, $f_{tj}/3.0$ for the BPELELU and BPELELUS options, and $0.16\sqrt{f_{cj}}$ for the BPHPELU and BPHPELUS options, according to Appendix 14 of BPEL dedicated to high performance concretes):

$$\tau_u - \delta\tau_u = k.\frac{\sin(\alpha+\beta_u)}{\sin\beta_u}\cdot x \qquad (12.21)$$

with the notations below, and others (extracted from BPEL):

- τ_u: tangential stress at the ultimate limit state, defined by the value of the component 1 of the no_effet effect;
- f_{tj}, f_{cj}: concrete strength in tension and compression (parameters v1 and v2);
- α: angle formed by steel bars with the beam mean fiber (between 45.0 and 90.0 degrees), calculated using the current value of the parameter v4;
- f_e: elastic limit of passive steel bars according to BPEL;
- f_{ywd}: elastic limit of passive steel bars according to EC2.2;
- γ_s: safety factor, for passive steel bars;
- A_t: sum of the areas of the cross sections of a set of passive reinforcing steel bars;
- s_t: spacing of these reinforcements, measured along the mean fiber of the beam;
- b_n: net thickness of a thin wall, in a tangential stress calculation point .

There are three possibilities:

- **in the absence of transverse prestress, and if b_n and $s_t$are constants,** the calculated value x corresponds to A_t and a coefficient k must be provided given by:

$$\begin{aligned} &BPEL \quad k = \frac{f_e}{\gamma_s \cdot s_t \cdot b_n} \Rightarrow résultat\ pcp : x = A_t \\ &EC2.2 \quad k = \frac{f_{ywd}}{s_t \cdot b_n} \Rightarrow résultat\ pcp : x = A_t \end{aligned} \qquad (12.22)$$

- **in the absence of transverse prestress, and if b_n or s_t are variables**, the calculated value x is the section of steel bars per unit length of the beam, and by net unit thickness of the thin wall; a corrected coefficient k must accordingly be provided:

$$\begin{aligned} &BPEL \quad k = f_e/\gamma_s \Rightarrow \quad résultat\ pcp : x = A_t/(s_t \cdot b_n) \\ &EC2.2 \quad k = f_{ywd} \Rightarrow \quad résultat\ pcp : x = A_t/(s_t \cdot b_n) \end{aligned} \qquad (12.23)$$

- **in the presence of transverse prestress,** the calculated value x represents the shear force to be balanced; a coefficient k equal to 1.0 must be provided and the part taken by the active steel bars, deducted from x.

$$\begin{array}{lll} BPEL & k = 1 \Rightarrow & résultat\ pcp : x = T \\ EC2.2 & k = 1 \Rightarrow & résultat\ pcp : x = T \end{array} \tag{12.24}$$

Conditions of use

- The Conditions of use of the TAULIMITES command apply to the ACIERS command.

Examples

```
$ processing of three study domains of a beam (command block)
$ domain 10, of which the two components are alternately studied,
$ covers the TANGENTIAL and NORMAL STRESSES, and certain sections
$ the ACIERS commands are not executed for study domains
$ 1 and 11, which are of other types
$ the transverse prestress exists and the minimum quantities of steels
$ are claculated with the same effect and according to two criteria
$ the definition parameters of the transverse steels are assumed to be
$ constant, and in particular, their axes are assumed to be parallel to
$ those of the thin walls produced by the generic sections
TRAITEMENT
  ETUDE  1
  ETUDE  10  COMPOSANTE  1  CONCOMITANTE  2
             COMPOSANTE  2  CONCOMITANTE  1
  ETUDE  11;
  ENVELOPPE  1000 ...
  ACIERS  500
  'ACIERS TRANSVERSAUX, COEFFICIENT betau CALCULE'
  BPELELU
  RESISTANCE  TRACTION  300.0  COMPRESSION  3000.0
  COEFFICIENT  1.0
  ACIERS  CONSTANTES  200.0  0.0
  ENVELOPPE  1000;
  ACIERS  501
  'ACIERS TRANSVERSAUX, COEFFICIENT betau FOURNI'
  BPELELUS  45.0
  RESISTANCE  TRACTION  300.0  COMPRESSION  3000.0
  COEFFICIENT  1.0
  ACIERS  CONSTANTES  200.0  0.0
  ENVELOPPE  1000;
  ..............
  EDITER  ACIERS  500;
  ENREGISTRER  ACIERS  500;
  EDITER  ACIERS  501;
  ENREGISTRER  ACIERS  501;
  ..............
FINTRAITEMENT;
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.11 – CDS_EC

CDS_EC LABEL_CALCUL label_calcul

$$\text{OPTION} \left\{\begin{matrix}\underline{\text{PONDERATION\_LOI}}\\ \text{RETOUR\_ETAT\_0}\end{matrix}\right\}$$

$$[\text{MATERIAU} \left\{\begin{matrix}\text{FICHIER nom\_fichier\_materiaux}\\ \underline{PCP}\end{matrix}\right\}]$$

$$[\text{FERRAILLAGE} \left\{\begin{matrix}\underline{\text{EXPLICITE}}\\ \text{AUTOMATIQUE}\end{matrix}\right\} \left\{\begin{matrix}\text{FICHIER nom\_fichier\_ferraillage}\\ \underline{PCP}\end{matrix}\right\}]$$

$$[\text{GEOMETRIE} \left\{\begin{matrix}\text{FICHIER nom\_fichier\_geometrie}\\ \underline{PCP}\end{matrix}\right\}]$$

$$[\left\{\begin{matrix}\underline{\text{JUSTIFICATION}}\\ \text{DIMENSIONNEMENT}\end{matrix}\right\} \text{DEFORMATIONS} \left\{\begin{matrix}\underline{BORNEES}\\ \text{NON\_BORNEES}\end{matrix}\right\}$$

$$\text{PIVOT\_C} \left\{\begin{matrix}\underline{\text{JUSTIFIE}}\\ \text{NON\_JUSTIFIE}\end{matrix}\right\} [\text{DIAGRAMME\_INTERACTION}]]$$

$$\begin{matrix}\text{SOLLICITATIONS}\\ **\end{matrix} \left(\begin{matrix}\text{ETAT LIMITE etat\_limite SITUATION situation} \left\{\begin{matrix}\text{PONDERES}\\ \text{A\_PONDERER}\end{matrix}\right\}\\ * \left\{\begin{matrix}\left\{\begin{matrix}G\\ GQ\end{matrix}\right\} \text{type\_effet no\_effet\_gq [STRUCTURE nom\_str\_gq]}\\ \left\{\begin{matrix}P\\ \text{GP}\end{matrix}\right\} \text{type\_effet no\_effet\_gp [STRUCTURE nom\_str\_gp]}\\ \text{TENSIONS STRUCTURE nom\_str\_p [RHO\_INJECTION rho]}\\ \text{[PONDERATION FAVORABLE k\_fav DEFAVORABLE k\_def]}\\ \text{Q type\_effet no\_effet\_q [STRUCTURE nom\_strp\_q][POLYEDRE facteur]}\end{matrix}\right\}\end{matrix}\right)$$

;

Parameters

- Label_calcul: calculation label that must identify its nature. This label will appear in the CDS file name and in the result files.
- nom_fichier_materiaux: name of the file, without the ".don" extension (full name: nom_fichier_materiaux.don), including the material parameters. This file, written by the user, includes the sub-command MATERIAU of CDS (see CDS user guide in annex) which are added to the generated CDS command file. By default, values of material calculated by PCP are used.
- nom_fichier_ferraillage.don: name of the file, without the ".don" extension (full name: nom_fichier_ferraillage.don), including the parameters of the FERRAILLAGE_AUTOMATIQUE. This file, written by the user, includes the sub-command FERRAILLAGE_AUTOMATIQUE of CDS (see CDS user guide in annex) which are added to the generated CDS command file. By default, default options of automatic reinforcement of CDS are used.

- nom_fichier_geometrie: name of the file, without the ".don" extension (full name: nom_fichier_geometrie.don), including the geometry definition of the section. This file, written by the user, includes the sub-commands of the command CONTOUR of CDS (see CDS user guide in annex) which are added to the generated CDS command file. By default, the geometry defined in PCP for the beam and the section is used.
- etat_limite: to be chosen among the followings :
  - ELS (FREQUENT/CARACTERISTIQUE/QUASIPERMANENT)
  - ELS_NON_FISSURE (FREQUENT/CARACTERISTIQUE/QUASIPERMANENT)
  - ELU (DURABLE/ACCIDENTEL)
- situation: to be chosen among the followings :
  - EXPLOITATION SERVICE
  - EXPLOITATION INFINIE
  - CONSTRUCTION
  - ACCIDENTELLE
  - SEISME
- type_effet, no_effet_gq et nom_strp_gq: parameters identifying the effect G or GQ to be considered as the loading to be applied to the sections studied by CDS (type_effet : see TRAITEMENT command).
- type_effet, no_effet_gp et nom_strp_gp: parameters identifying the effect P or GP to be considered as the loading to be applied to the sections studied by CDS (type_effet : see TRAITEMENT command).
- nom_str_p: name of the saved structure (by the PH3 module) identifying a state from which the tension of the prestressed cables and their grouting state are taken.
- rho: participation factor of the prestressing to the balancing section (between 0 and 1 included). Its default value is 1. This factor is applied to cables with grouting only.
- k_fav, k_def: ponderation factor of the tensions of the prestressing cables calculated by PCP. These factors have to be set only if the option PONDERE is chosen; otherwise, CDS will do the necessary ponderation.
- facteur: ponderation factor applied on the load linked to the effect no_effet_q when the option POLYEDRE is used.

Functions

This command is to dimension or justify under normal stresses sections of an area of study of EFFORTS SECTIONS or EFFORTS ELEMENTS type, studying the combined asymmetrical bending components (FSX, MSY and MSZ, see Table 1.1) with the CDS software. In the case of EFFORTS SECTIONS type, only the contoured sections can be used in this command. The dimensioning is available only if ther is reinforcing steel in the section (see FERRAILLAGE option). This command is useful only if the user has at least the 6.xx version of CDS.

For each studied section, the ETU module create the commands to define concrete, outline, potential prestressing, and loading andcontrol commands in a file identified by the label of the CDS calculation (label_calcul), and by the element number. The file appears in the CDS subdiretory of the current case.

After the ETU module execution, one must execute CDS under PCP by selecting the "Cds_Exe_label_calcul.don" file automatically generated by PCP, or individualy use the CDS data files of the CDS directory. In the first cas, the user has to write in the "Setconf.tcl" file, in the PCP install directory, the path to the executable file of CDS. All the sections are processed then by CDS. Computation reports appear in the CDS dorectory of the current case.The Cds_Exe_label_calcul.prt file includes all the computation reports and the summary of the computation.

Materials

The characteristics of the materials are either determined by PCP from the data introduced by the user when they are defined in PCP, or redefined and supplemented in the nom_fichier_materiau.don file whose content is integrated into the CDS command created.

With the PCP option, ETU will calculate the characteristic strengths of the materials from the elasticity module of the materials defined in data of the modules defining the geometry of the POUTRES. It will determine the options of the MATERIAUX from the features defined in PCP and for the other options, these are those taken by default by CDS. It is therefore up to the user to verify that they correspond to the project.

If the user wants to introduce more parameters (partial coefficients for boundary states, etc.), they will have to create the nom_fichier_materiaux.don file and incorporate CDS Advanced Material Definition Commands (see CDS notice). The materials will then have to be defined with the same names as those used by PCP if they already exist. This is particularly the case for concrete and pre-stress.

Reinforcement

The reinforcement can be defined in four ways:

EXPLICITE reinforcement defined in PCP (see definition of POUTRES): either the user decides to redefine the characteristics in the MATERIAU file: he then takes the same name as PCP to redefine its characteristics in the "nom_fichier_materiaux.don" file, or ETU deducts the characteristics of the steel parameters defined when defining the POUTRES.

EXPLICITE reinforcement defined in the "nom_fichier_ferraillage.don" file: The user must then define the material in the "nom_fichier_materiaux.don" file with the same steel name in both files. The "nom_fichier_ferraillage.don" file then contains the explicit definition commands of a reinforcement (see CDS notice)

AUTOMATIQUE PCP reinforcement: ETU will produce automatic FERRAILLAGE options for CDS, but an ACIER material must be defined in the "nom_fichier_materiaux.don" file and the name given to this material must be: aci_pcp. The options of AUTOMATIC FERRAILLAGE for CDS are: EXTRADOS and INTRADOS scrap on a single bed with the default diameters of CDS.

AUTOMATIQUE reinforcement defined by nom_fichier_ferraillage.don: the ACIER materials defined in the "nom_fichier_materiaux.don" file and in the nom_fichier_ferraillage.don file must be identical. The automatic reinforcement definition commands in the "nom_fichier_ferraillage.don" file are those of CDS (see Notice CDS):

Prestressing

The method of calculating pre-stress sections PONDERATION_LOI corresponds to the method of weighting the law of CDS. The RETOUR_ETAT_0 method corresponds to the CDS return to state 0 method.

ETU places itself in the hypothesis of the POST-TENSION and therefore simultaneously applies G and P and then applies Q. The GQ option is therefore prohibited in pre-stressed concrete.

The rho coefficient depends on the rule of consideration of surges in cables. Either the surge should not be considered and in which case rho is to be taken zero, or it must be fully considered and in which case one fixes rho equal to 1. A partial surge is taken into account by setting a rho value between 0 and 1. For cables NON_INJECTES ETU systematically fixes rho to zero.

ETU takes into account the state of the wiring defined at the time of the STRUCTURE SAUVEGARDE nom_str_p by PH3. Only INTERIEURS cables are provided for CDS. As a result, external cables are ignored. The INJECTE or non-INJECTE status is also automatically defined by PCP based on the situation when the STRUCTURE nom_str_p is saved by PH3.

Solicitations

The limit states of the type ELS and ELU DURABLE concern exclusively the situations EXPLOITATION or CONSTRUCTION. The ACCIDENT ELU deals exclusively with the SEISM and ACCIDENT situations.

The applied loads are the result of a permanent G-pressure state, to which a possible P effect is superimposed and a Q load, considered a simple or Gaussian load (see POLYEDRE):

The G effort is the result of a simple ETAT, an ETAT PONDERE or an ETAT EXTREMAL from PH3.

The P efforts are the result of an ETAT PONDERE or an ETAT EXTREMAL from PH3 that encompass the ISO and HYPER effects of pre-stress. P can be isolated or combined with G (GP option) with the PONDERES option.

The Q effort is usually the result of an EFFET created under ETU by combination or envelope.

Pre-stress implementation data (stretched or unstretched cables, injected or unstretched cables, likely voltages in the cables) is produced from the backed-up structure "nom_str_p" by PH3. The injection coefficient only applies to cables injected at the time of the backup of the "nom_str_p" STRUCTURE by PH3. For non-injected cables, ETU takes zero value for this coefficient.

These efforts can be previously PONDERES by regulatory coefficients or to PONDERER by CDS. In the first case they can be defined grouped (GP, GQ) or separated. In the second case they can only be defined separately so that CDS can apply the default coefficients.

The tables below illustrate the different scenarios:

Type effort	G	P	GP	Q
PCP calculations	In PH3 is a ETAT PONDERE with KGxx=1 and KPxx=0	In PH3 is a ETAT PONDERE with KGxx=0 and KPxx=1	Prohibited	In ETU is a: Combination, Envelope, Equity Weighting Q
Transfer to CDS	Likely G-state	Likely G-State and probable tensions		Q max and min effect
CDS	G weighting calculation:0.95,1.05 in ELS and 1.35 in ELU	P:0.90,1.10 weighting calculation at ELS and 1.00 in ELU		Calculation with weighting of Q:1 to ELS and 1.2 to DURABLE ELU and 1 to ACCIDENTEL ELU.

Table 12.2: CDS A_PONDERER solicitation cases

Type effort	G	P	GP	Q
PCP calculations	In PH3 is an ETAT PONDERE with: KGxx=0.95,1.05 to ELS or 1.1.35 to ELU KPxx=0	In PH3 is an ETAT PONDERE with: KPxx=0.90, 1.10 in ELS and 1 in ELU KGxx=0	In PH3 is an ETAT PONDERE with KGxx=0.95, 1.05 aux ELS and 1, 1.35 aux ELU KPxx=0.90,1.10 to ELS or 1 to ELU	In ETU is a: Q Stock Combination, Envelope, or Weighting
Transfer to CDS	Weighted state G max and min	Weighted state P max and min and tensions affected by k_fav and k_def.	Weighted state GP max and min and tensions affected by k_fav and k_def.	Q max and min effect
CDS	Calculating without G weighting	P-weighted calculation	Calculating without GP weighting	Calculation without Q weighting

Table 12.3: solicitation cases already PONDEREES by PCP

CDS weighting if A_PONDERER

Limit state	G	P	Q
ELS	0.95/1.95	0.90/1.10	0/1
SUSTAINABLE ELU	1/1.35	1	0/1.2
ACCIDENTAL ELU	1	1	0/1(*)

(*)Wrong value for the wind: you have to switch to PONDERES mode for a wind-type Q charge.

Polyhedre option

With the POLYEDRE EFFET option, the no_effet effect is considered as a Gaussian-type load, to be added, after weighting by the factor parameter, to the permanent G-P state; 24 loads are built by considering the peaks of the polyhedra confined to the ellipsoid of this Gaussian type load. This method is documented in "Alain CAPRA et Victor DAVIDOVICI : Calcul dynamique des structures en zone sysmique Eyrolles 1980".

The following treatments are performed:

Search for the 9 characteristic values of the Gaussian-type effect (the diagonal terms of the corresponding matrix are the standard deviations-of the components considered, and the crosswords are their concomitant components):

$$\begin{bmatrix} FSX & MSY_{FSX} & MSZ_{FSX} \\ FSX_{MSY} & MSY & MSZ_{MSY} \\ FSX_{MSZ} & MSY_{MSZ} & MSZ \end{bmatrix} \tag{12.25}$$

Establishing the corresponding covariance matrix (always symmetrical for a Gaussian-type effect):

$$\begin{bmatrix} FSX^2 & FSX \cdot MSY_{FSX} & FSX \cdot MSZ_{FSX} \\ MSY \cdot FSX_{MSY} & MSY^2 & MSY \cdot MSZ_{MSY} \\ MSZ \cdot FSX_{MSZ} & MSZ \cdot MSY_{MSZ} & MSZ^2 \end{bmatrix} \tag{12.26}$$

Determining the 24 triplets of values*(FSX,, MSY,, MSZ)*representing the tops of the polyhedra circumscribed to the equiprobability ellipsoid, defined by the covariance matrix; if one projects the ellipsoid on a plane of axes ox and oy representing a couple of values extracted from a triplet, one obtains four peaks of the polydre:

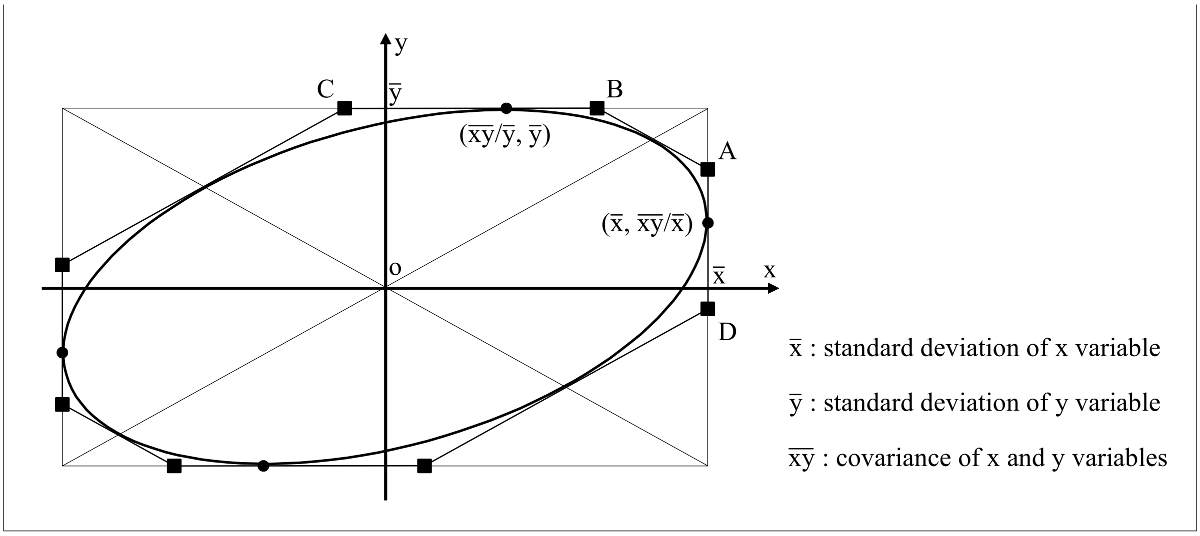


Figure 12.4 - Equiprobability Ellipsoid, four peaks of the circumscribed polyhedra

$$A\begin{cases}\bar{x}\\ \bar{y}\left[\sqrt{2(1+\rho)}-1\right]\end{cases} \qquad B\begin{cases}\bar{x}\left[\sqrt{2(1+\rho)}-1\right]\\ \bar{y}\end{cases} \tag{12.27}$$

$$C\begin{cases}\bar{x}\left[1-\sqrt{2(1-\rho)}\right]\\ \bar{y}\end{cases} \qquad D\begin{cases}\bar{x}\\ \bar{y}\left[1-\sqrt{2(1-\rho)}\right]\end{cases} \tag{12.28}$$

$$\rho = \frac{\bar{x}\bar{y}}{\bar{x}\cdot\bar{y}} \tag{12.29}$$

The ellipsoid is projected successively on each of the three possible planes; each side of its circumscribed parallelepiped providing four peaks of the polyhedra, one obtains the 24 peaks sought; each triplet representing a summit of the polyhedra is then weighted by the factor parameter and added to the permanent stresses, to produce a load.

Sizing

Sizing only applies with the FERRAILLAGE option. In the case of a AUTOMATIQUE FERRAILLAGE, CDS will determine the position and number of steels on each layer.

In the case of an EXPLICITE reinforcement, the sizing, if necessary, leads CDS to increase the overall amount of steel in each area: INTRADOS or EXTRADOS.

The sizing is systematic followed by a Justification.

Justifications

The justifications apply to steels already present in the PCP sections or to steels from the DIMENSIONMENT previously made.

With the option DEFORMATIONS BORNEES the balance of the section is sought by limiting itself to the licit deformations of the borderline state treated. With the DEFORMATIONS option NON_BORNEES the balance is sought regardless of the limits of the licit deformations but the software signals overruns.

The option on pivot C, allows the user to ask for its justification.

Conditions of use

- This optional order can be introduced several times during a processing.
- This command is prohibited when the area of study to which it applies is not of the EFFORTS SECTIONS type, or when the FSX, MSY and MSZ components are not studied simultaneously.
- CdS files are created in the case's CDS directory. The user can launch CDS by selecting each file.
- The "Cds_Exe_Label_calcul.don" file allows you to launch all the CDS.
- With the POLYEDRE EFFET options, it is not verified that the covariance matrix is symmetrical.
- Options A_PONDERER are prohibited for GP-type efforts.
- If a COMBINAISON or an ENVELOPPE QUADRATIQUE is used it must be ENREGISTREE and carried out in a prior TRAITEMENT and FINTRAITEMENT.
- With the POLYEDRE option, if a combination is necessary to evaluate the total Q effect, this combination must be performed successively on each of the study components with their concomitant and recorded in a prior PROCESSING command (see earthquake example below).

Methodological advice

- The POLYEDRE option is to be used with Q actions corresponding to a SPECTRAL REPONSE of the type VENT or the type SISMIQUE.
- For a wind study at the ELU, the weighting 1.2 on Q must be introduced in PONDERE mode because the default of CDS is 1.
- For sizing, it is advisable to group all the limit states into a single CDS control in order to lead to an overall optimum sizing.
- In the case of an earthquake, a quadratic combination of the different seismic directions is exclusive of an application of the Eurocode accumulation rule using the coefficients 0.3 striking the concomitant directions. To apply this rule of accumulation, a traditional COMBINATION will be used (cf. example earthquakes on current elements).

Examples

```
$ Example of the encorb_sci case

$ The study below focuses on the EFFORTS in SECTIONS systems in a beam
part; $ the three normal solicitation components are dealt with in
succession:
$ Q charges are defined by envelopes
$ The material is defined in the mate_cds.don file
$ Automatic reinforcement is defined in the fer_cds.don file
$ The GP state is obtained by the command ETAT PONDERE defined in the
$ phasing
$
PCPETU ;
TITRE 'ETUDE DU PONT - justification des sections en CARA et ELU';
TRAITEMENT ETUDE EFFORTS SECTIONS
COMP 1 CONC  5 6
```

```
COMP 5 CONC  1 6 ;
ENVELOPPE   400 ' GRADIENT THERMIQUE 10 DEGRES'
COEF 0.  CHARGE 20 STRUCTURE STRUSERV
COEF 1.  CHARGE 20 STRUCTURE STRUSERV ;
COMBINAISON 500 'Q  CARA : LM1,  TROTTOIR , GRADIENT 5 DEGRES '
COEF 1.0   ACTION 30 STRUCTURE STRUSERV
COEF 1.0   ACTION 40 STRUCTURE STRUSERV
COEF 0.5   ENVELOPPE 400;
COMBINAISON 600 'Q  ELU : LM1,  TROTTOIR '
COEF 1.2   ACTION 30 STRUCTURE STRUSERV
COEF 1.2   ACTION 40 STRUCTURE STRUSERV;

CDS_EC
LABEL_CALCUL            ELS_ELU
OPTION                  RETOUR_ETAT_0
MATERIAU FICHIER        mate_cds $ material redefinition
FERRAILLAGE AUTOMATIQUE   FICHIER fer_cds  $ automtic reinforcement
                                           $ intrados et extrados
DIMENSIONNER DEFORMATION BORNEES  PIVOT JUSTIFIER DIAGRAMME
SOLLICITATIONS
$ ELS CARACTERISTIQUES
ETAT_LIMITE ELS CARACTERISTIQUE  SITUATION  EXPLOITATION SERVICE  PONDERES
         GP ETAT 100  STRUCTURE STRUSERV
         TENSIONS STRUCTURE STRUSERV RHO_INJECTION 1
         PONDERATION FAVORABLE 0.90 DEFAVORABLE 1.10
         Q  COMBINAISON  500
 ETAT_LIMITE ELS CARACTERISTIQUE  SITUATION  EXPLOITATION INFINI  PONDERES
         GP  ETAT 100  STRUCTURE STRUINFI
         TENSIONS STRUCTURE STRUINFI RHO_INJECTION 1
         PONDERATION FAVO 0.90 DEFA 1.10
         Q  COMBINAISON  500
$ ELU
ETAT_LIMITE ELU DURABLE  SITUATION  EXPLOITATION SERVICE  PONDERES
         GP  ETAT 200  STRUCTURE STRUSERV
         TENSIONS STRUCTURE STRUSERV RHO_INJECTION 1
         PONDERATION FAVO 1 DEFA 1
         Q COMBINAISON 600
ETAT_LIMITE ELU DURABLE  SITUATION  EXPLOITATION INFINI  PONDERES
         GP  ETAT 200  STRUCTURE STRUINFI
         TENSIONS STRUCTURE STRUINFI RHO_INJECTION 1
         PONDERATION FAVO 1 DEFA 1
         Q  COMBINAISON  600 ;
FINTRAITEMENT ;
FIN ;
```

```
Phasing file:
PHASAGE
```

```
----------
CALCUL EXTREMAS EFFORTS SECTIONS COMP 1 CONC 5 4 6
CALCUL EXTREMAS EFFORTS SECTIONS COMP 5 CONC 1 4 6
CALCUL EXTREMAS EFFORTS SECTIONS COMP 6 CONC 1 4 5
------------
$
SAUVE STRUSERV
TITRE 'ETAT PONDERE CARACTERISTIQUE EN SERVICE'
ETAT PONDERE KGFA 0.95 KGDE 1.05 KPTFAVORABLE 0.90 KPTDEFAVORABLE 1.10 100
TITRE 'ETAT PONDERE ELU EN SERVICE'
ETAT PONDERE KGFAVORABLE 1 KGDEFAVORABLE 1.35 200
TITRE 'OUVRAGE AU TEMPS INFINI A VIDE'
DATE VIEILLISSEMENT INFINI ETAT
SAUVE STRUINFI
TITRE 'ETAT PONDERE CARACTERISTIQUE INFINI'
ETAT PONDERE KGFA 0.95 KGDE 1.05 KPTFAVORABLE 0.90 KPTDEFAVORABLE 1.10  100
TITRE 'ETAT PONDERE ELU INFINI'
ETAT PONDERE KGFAVORABLE 1 KGDEFAVORABLE 1.35 200
FIN

Fichier mate_cds.don :
MATERIAU BETON BETON  $ meme nom que materiau PCP
     TITRE                          ' Materiau beton fck =   43.55     '
     FCK                             43.55
     SECTION                         CAISSON
     EXPOSITION                      XS1
     PRECONTRAINT                    oui
     CIMENT                          R
     FATIGUE_CRITERE_SIMPLIFIE       TRACTION OUI
 FIN_MATERIAU
 MATERIAU PRECONTRAINTE CABTYP1  $ meme nom que cable PCP
     TITRE                       "CABTYP1 "
     FP01K                       (0.9*1770/0.8)
     FPK                           (1770/0.8)
     TYPE                         TORON
 FIN_MATERIAU
  MATERIAU PRECONTRAINTE CABTYP2 $ meme nom que cable PCP
     TITRE                       "CABTYP2 "
     FP01K                         (0.9*1777/0.8)
     FPK                           (1777/0.8)
     TYPE                         TORON
 FIN_MATERIAU
 MATERIAU ACIER ACI_PCP $ nom repris dans fer_cds.don
     TITRE                        "ACIPCP "
     FYK                          500.0
     TYPE                         T_A
     SECTION                      CAISSON
     CONFINE                      OUI
     FATIGUE_CRITERE_SIMPLIFIE    OUI
     FISSURATION_CRITERE_SIMPLIFIE OUI
```

```
 FIN_MATERIAU

Fichier de definition du ferraillage fer_cds.don
        FERRAILLAGE fer_pcpsup
             TITRE    "fersup"
             MATERIAU aci_pcp
             OPTIONS_FERRAILLAGE_AUTOMATIQUE
             DISPOSITION SUPERIEURE STRICTE
             FIN_OPTIONS
        FIN_FERRAILLAGE
        FERRAILLAGE fer_pcp_inf
             TITRE    "ferinf"
             MATERIAU aci_pcp
             OPTIONS_FERRAILLAGE_AUTOMATIQUE
             DISPOSITION INFERIEURE STRICTE
             FIN_OPTIONS
         FIN_FERRAILLAGE


$ Example of the encorb_sci case with polyhedra effect on seism.
$ The study below focuses on the EFFORTS in SECTIONS systems in a beam
$ part; the three normal solicitation components are dealt with in
$ succession:
$ Q charges are defined by seismic responses
$ the material is deduced from PCP
$ There is no sizing and no reinforcement
$ The GP state is the same as previously
$
PCPETU;
TITRE 'ETUDE DU PONT - justification des sections au seisme';
$ construction de la combinaison Quadratique
TRAITEMENT ETUDE EFFORTS SECTIONS
  COMP 1 CONC  5 6
  COMP 5 CONC  1 6
  COMP 6 CONC  1 5;
COMBINAISON QUADRATIQUE 2000  "Combinaison seisme vertical/horizontal"
COEF 0.5 REPONSE   1001  STRUCTURE STRUSERV
COEF 0.5 REPONSE   1002  STRUCTURE STRUSERV
ENREGISTRER  COMBINAISON 2000;
FINTRAITEMENT;

$ Justification to seism
TRAITEMENT ETUDE EFFORTS SECTIONS
  COMP 1 CONC  5 6
  COMP 5 CONC  1 6
  COMP 6 CONC  1 5;
CDS_EC
      LABEL_CALCUL SEISME
      OPTION                           RETOUR_ETAT_0
      MATERIAU                         PCP
      JUSTIFIER DEFORMATION BORNEES  PIVOT JUSTIFIER DIAGRAMME
      SOLLICITATIONS
         $ ELU a la mise en service
```

```
         ETAT_LIMITE ELU ACCIDENTELLE  SITUATION SEISME  PONDERES
         GP  ETAT 200  STRUCTURE STRUSERV
         TENSIONS STRUCTURE STRUSERV RHO_INJECTION 1
         PONDERATION FAVO 1 DEFA 1
          Q   REPONSE       1001  STRUCTURE STRUSERV  POLYEDRE 1.2
          Q   REPONSE       1002  STRUCTURE STRUSERV  POLYEDRE 1.2
          Q   COMBINAISON   2000                POLYEDRE 1.2 ;
FINTRAITEMENT ;
FIN ;

$ Commandes du fichier seisme :
REPONSE MODALE SEISME
'seisme transversal'
COMPOSANTE Y SUPERPOSITION CQC ENREGISTRER 1001 ;

REPONSE MODALE SEISME
'seisme vertical'
COMPOSANTE Z SUPERPOSITION CQC ENREGISTRER 1002 ;


$ Example of the polyhedra effect on seism for current éléments with
Eurocode combinations of the three seism directions : 9100, 9200 et 9300.
$ Material and reinforcement are deduced from PCP.
$ Geometry of the section is defined in the contoured section file.
$ The state GP is the same than before.
$
PCPETU;
TITRE 'ETUDE DU PONT - justification des sections au seisme';
$ construction de la combinaison des trois directions du seisme
TRAITEMENT ETUDE EFFORTS SECTIONS
COMP 1 CONC  5 6 $ mandatory for the polyedre option
COMP 5 CONC  1 6 $ mandatory for the polyedre option
COMP 6 CONC  1 5; $ mandatory for the polyedre option
COMBINAISON  9501 'Seism Ex -+ Sx +- 0.3Sy +- 0.3Sz'
FAVORABLE  -1      DEFAVORABLE 1     REPONSE 9100 STRUCTURE STRUSERV
FAVORABLE  -0.3   DEFAVORABLE 0.3    REPONSE 9200 STRUCTURE STRUSERV
FAVORABLE  -0.3   DEFAVORABLE 0.3    REPONSE 9300 STRUCTURE STRUSERV ;

COMBINAISON  9502 'Seism Ex -+0.3 Sx +- Sy +- 0.3Sz'
FAVORABLE  -0.3   DEFAVORABLE 0.3    REPONSE 9100 STRUCTURE STRUSERV
FAVORABLE  -1      DEFAVORABLE 1     REPONSE 9200 STRUCTURE STRUSERV
FAVORABLE  -0.3   DEFAVORABLE 0.3    REPONSE 9300 STRUCTURE STRUSERV;

COMBINAISON  9503 'Seism Ex -+ 0.3Sx +- 0.3Sy +- Sz'
FAVORABLE  -0.3   DEFAVORABLE 0.3    REPONSE 9100 STRUCTURE STRUSERV
FAVORABLE  -0.3   DEFAVORABLE 0.3    REPONSE 9200 STRUCTURE STRUSERV
FAVORABLE  -1     DEFAVORABLE 1      REPONSE 9300 STRUCTURE STRUSERV;

ENVELOPPE  9550 'Ex-Ey-Ez'
COEFFICIENT 1 COMBINAISON 9501
COEFFICIENT 1 COMBINAISON 9502
COEFFICIENT 1 COMBINAISON 9503;
```

```
EDITER ENVELOPPE 9550;
ENREGISTRER  ENVELOPPE 9550;
FINTRAITEMENT; FINTRAITEMENT;

$Justification
TRAITEMENT ETUDE EFFORTS SECTIONS
  COMP 1 CONC  5 6; $ the definition of the torseur is enough because PCP
                    $ will used the other components to generate all the
                    $ polyedra.

CDS_EC
      LABEL_CALCUL SEISME
      OPTION                             RETOUR_ETAT_0
      MATERIAU                           PCP
      GEOMETRIE                          FICHIER contour
      FERRAILLAGE                        EXPLICITE PCP
      JUSTIFIER DEFORMATION BORNEES  PIVOT JUSTIFIER DIAGRAMME
      SOLLICITATIONS
         $ ULS
         ETAT_LIMITE ELU ACCIDENTELLE  SITUATION SEISME  PONDERES
         GP  ETAT 200  STRUCTURE STRUSERV
         TENSIONS STRUCTURE STRUSERV RHO_INJECTION 1
         PONDERATION FAVO 1 DEFA 1
         Q   ENVELOPPE   9550     POLYEDRE 1 ;
FINTRAITEMENT ;
FIN ;

Fichier Contour.don

       TITRE " Contoured section"
       POINT
          NUMERO  1  Y  1.500  Z   0.100
          NUMERO  2  Y -1.500  Z   0.100
          NUMERO  3  Y -1.500  Z   1.500
          NUMERO  4  Y -2.000  Z   1.500
          NUMERO  5  Y -2.000  Z  -1.500
          NUMERO  6  Y -1.500  Z  -1.500
          NUMERO  7  Y -1.500  Z  -0.100
          NUMERO  8  Y  1.500  Z  -0.100
          NUMERO  9  Y  1.500  Z  -1.500
          NUMERO 10  Y  2.000  Z  -1.500
          NUMERO 11  Y  2.000  Z   1.500
          NUMERO 12  Y  1.500  Z   1.500
       FIN_POINT
       $ Potential holes in the section
```

Linked commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.12 - COMMENTS

COMMENTAIRE texte_info ;

Parameters

- texte_info: informative text (string);

Functions

This command causes the insertion of an informative text line in the results file.

Conditions of use

- This optional command can be introduced several times during processing.

Examples

```
$ insertion of a line of underlined text
COMMENTAIRE
'CALCUL DES SOLLICITATIONS ENVELOPPES, SELON LE BPEL'
COMMENTAIRE
'----------------------------------------------------'
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.13 - EDITER

EDITER [**A4**]
type_effet no_effet [STRUCTURE nom_structure] ;

Parameters

- type_effet, nom_effet, nom_structure: parameter identifying the effect to be edited (see TRAITEMENT command).

Functions

This command causes the editing of an effect.

With the A4 option, results are produced over 80 columns maximum; by default, they are 132 columns maximum.

Conditions of use

- This optional command can be introduced several times during processing.
- Edited effects accumulate in the results file in the order of introduction: the EDITER commands for an area of study, areas of study for processing, and processing for a session.
- All types of effects can be edited, including the terminal types (see TRAITEMENT command).
- The effect to be edited must exist in the memory or database.

Examples

```
$ effects created by modules PH3 and ENV
$ and present in the database
EDITER  CHARGE  100;
EDITER  ETAT  0  STRUCTURE  STRUC1;
EDITER  ACTION  100  STRUCTURE  STRUC1;

$ effects created by the ETU module
$ and present in memory or in the database
EDITER  ENVELOPPE  1000;
EDITER  COMBINAISON  100;
$ these effects will not be calculated, and therefore produced
$ only for the study domains in question
EDITER  TAULIMITES  51;
EDITER  ACIERS  151;
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.14 - EXPORT

$$\text{EXPORTER} \quad \begin{Bmatrix} \text{GUIVIRGULES} \\ \text{VIRGULES} \\ \text{ESPACES} \\ \text{PCP} \end{Bmatrix} \quad \begin{Bmatrix} \text{MAX} \\ \text{MIN} \\ \text{MAX\_MIN} \end{Bmatrix}$$

type_effet no_effet [STRUCTURE nom_structure] [FICHIER nom_fichier];

Parameters

- type_effet, nom_effet, nom_structure: parameters identifying the effect to be exported (see TRAITEMENT command).
- nom_fichier: file name (to be placed between quotes) in which the effect will be exported. If the FICHIER option is not used, the file will have the same name as the data file in which the EXPORTER command is used, and will have the extension 'exp'.

Functions

This command causes the export of an effect, in a readable and modular form.

With the GUIVIRGULES option, the edited values are placed individually between quotes, and separated by commas.

With the VIRGULES or ESPACES options, values are output in "as is" form, and separated by commas (format compatible with MS™-EXCEL® spreadsheets), or spaces.

With the PCP option, the values are edited so that they can be imported into another case using the IMPORTER command.

MAX, MIN and MAX_MIN respectively allow you to edit only the maximum values, only the minimum values, or the maximum and minimum values.

Conditions of use

- This optional command can be introduced several times during processing.
- All types of effects can be exported, including the terminal types (see TRAITEMENT command).
- The effect to be exported must exist in the memory or database.
- Exported effects accumulate in the export file ("reset" at the beginning of each session) in the order of introduction: EXPORTER commands for an area of study, areas of study for processing, and processing for a session.

Methodological advice

- Avoid mixing EXPORTER and CDS commands in a single session.

Examples

```
EXPORTER  GUIVIRGULES  MAX_MIN  CHARGE  100;
EXPORTER  VIRGULES  MAX  COMBINAISON  1000;
```

```
EXPORTER  ESPACES  MAX_MIN  ACTION  100  STRUCTURE  STRUCT1;
EXPORTER  PCP  MAX_MIN ENVELOPPE 2000 FICHIER 'enve2000.exp';
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT ; IMPORTER ;

MS™ is a registered trademark of Microsoft Corporation; EXCEL® is a product of Microsoft Corporation

12.15 - IMPORT

IMPORTER nom_fichier type_effet no_effet ;

Parameters

- type_effet, nom_effet: parameters identifying the effect to be imported (see TRAITEMENT command).
- filename: file name (to be placed between quotes) in which the effect will be sought.

Functions

This command causes the import of an effect, and its database record.

Conditions of use

- This optional command can be introduced several times during processing.
- All types of effects can be imported, including the terminal types (see TRAITEMENT command).
- The numbers of the case studies in which effects are imported should correspond with the numbers of the case studies in which the EXPORTER command was used. Similarly, components and requested concomitants must match.

Examples

```
IMPORTER  'enve2000.exp'  ENVELOPPE 2000;
IMPORTER  'export.exp'  CHARGE 10;
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT ; EXPORTER ;

12.16 - SAVE

ENREGISTRER type_effet no_effet ;

Parameters

- type_effet, no_effet: parameters identifying the effect to be saved (see TRAITEMENT command).

Functions

This command causes an effect to be saved in the database, an effect that can be reused by the ETU module in subsequent processing of the current session or in a subsequent session, or viewed by the RES module.

Conditions of use

- This optional command can be introduced several times during processing.
- Only the effects created by the ETU module can be saved, including terminal effects (see TRAITEMENT command).
- The effect to be saved must have been created during the current processing.

Examples

```
ETUDES;
TRAITEMENT  $ process 1
  ETUDE  ...
  ..............
  COMBINAISON  1000 ...
  ..............
  ENVELOPPE   1001
  ..............
  ENREGISTRER  COMBINAISON  1000;
  ENREGISTRER  ENVELOPPE  1001;
FINTRAITEMENT;
TRAITEMENT  $ process 2
$ the combination of 1000 and the enclosure 1001 can be reused here
  ETUDE  ...
  ..............
  ENVELOPPE  1002;
  'TITRE'
  COEFFICIENT  1.0  COMBINAISON  1000
  COEFFICIENT  1.0  ENVELOPPE  1001;
  ..............
FINTRAITEMENT;
FIN;
```

Related commands

ETUDES ; TRAITEMENT ; FINTRAITEMENT

12.17 - FINTRAITEMENT

FINTRAITEMENT;

Functions

This order marks the end of the current processing (control block having started after the last TRAITEMENT command).

All effects created during this processing that have not been recorded will be erased from memory.

Conditions of use

- Each TRAITEMENT command must match a FINTRAITEMENT command.
- The nesting of processing sequences is prohibited.

Examples

```
ETUDES;
$ group 1
$ --------
TRAITEMENT
$ study domains and components
  ..............
$ miscellaneous calculations and commands:
$ LECTURE, COMBINAISON, PONDERATION, ENVELOPPE, TAULIMITES, ACIERS, CDS
  ..............
$ control of results and commands:
$ COMMENTAIRE, EDITER, EXPORTER, ENREGISTRER
  ..............
FINTRAITEMENT;
$ group 2
$ --------
TRAITEMENT
  ..............
  ..............
FINTRAITEMENT;
$ group 3
$ --------
TRAITEMENT
  ..............
  ..............
FINTRAITEMENT;
FIN;
```

Related commands

TITRE ; RAPPELER ; TRAITEMENT ; LECTURE ; COMBINAISON ; PONDERATION

ENVELOPPE ; TAULIMITES ; ACIERS ; CDS ; COMMENTAIRE ; EDITER

EXPORTER ; ENREGISTRER

12.18- END

FIN;

Functions

This command causes the end of a session and stops the execution of the ETU module; all potential following commands are ignored.

Examples

```
ETUDES;
.......
.......
FIN;
```

Related commands

ETUDES

Chapter 13

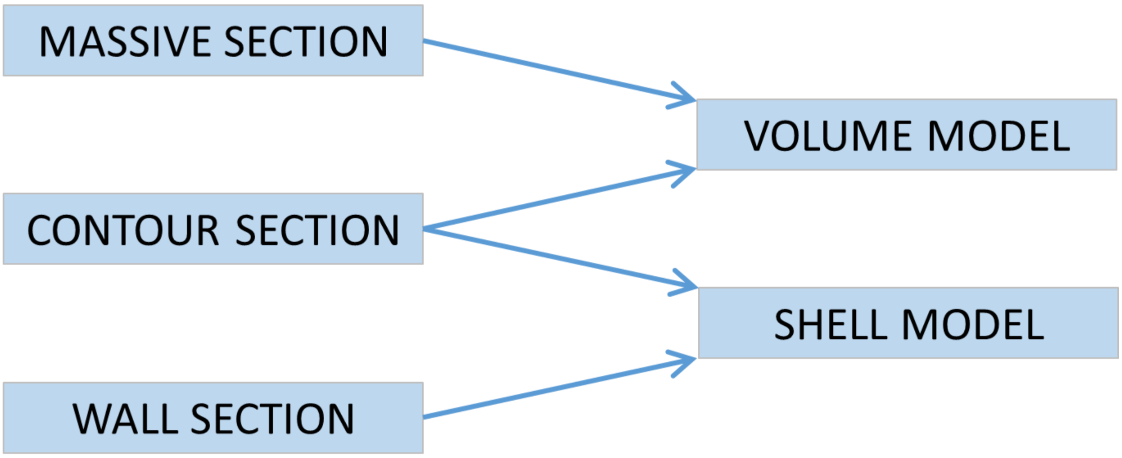
Finite elements

CONTENTS

Introduction

The purpose of this modulus is to give the opportunity to make a study in a finite elements computation software without redefining a new geometric model. This is possible thanks to the functioning of PCP, whose input data do not represent equivalent characteristics of the bars, but the real geometry of the bridge. As it calls a saved structure, this module has to be called after the phase modulus PH3.

The MAI modulus generates beam elements, shell/plate elements and volume elements. The generation of this modeling depends on the chosen method to describe the geometry with PCP. If the model is entirely defined with the SECTION TYPE command, only a model with beam elements can be generated. If one or several GE1 files have been defined, then shell and volume models can be generated according to the following compatibility scheme:



For a given model, the MAI modulus links two transversal sections point-to-point in the case of two identical sections, and use a closest point method in the case of two different sections. However, if two successive sections are too different, the MAI modulus cannot properly link the sections. In that case, the user has to define a short length transition element, which had to be modeled with a beam formulation.

It is also possible to generate hybrid models, with two or three different formulation. The MAI modulus automatically links thus the volume and beam formulations or surface and beam formulations. The link between 2D and 3D models is not automatically done yet.

This modulus generates a geometry file and a mesh file. If the structure has prestressed cables, meshes to link the concrete and the cables are created, and the new file including this links is named geometrie_liaisons. At this time, mesh file is available in .msh format. For this, the user must install the software GMSH.

A command file for finite element computation is also generated for the finite element software chosen by the user. At this time, the generated command file is relative to Code_Aster.

All the generated files are in a folder entitled gmsh, in the case directory.

Contents

13.1 – DISCRETISATION

DISCRETISATION [FORMAT format] [FICHIER $\left\{\begin{array}{l}\text{UNIQUE}\\ \hline \text{MULTIPLES}\end{array}\right\}$];

Parameters

- format: output format of the mesh file. At this time, mesh file is available in MSH or MED format. The current version of PCP can only generate a model with prestressing with the .msh format if the user has installed GMSH.

Functions

This command defines the beginning of the data for the finite element mesh. It defines also the mesh output format and the functioning mode of the MAI modulus.

With the MULTIPLES option, a segment is created only if both transversal sections are identical. At each discontinuity, a new file is created. With this mode, it is impossible to mix several formulation in one file.

With the UNIQUE option, the geometry of all sections, for any formulation, is gather in one file. If two successive transversal sections with the same formulation are different, the link is forced by a closest point method.

With the UNIQUE option, it is also possible to mix several formulation (1D/2D/3D). Besides, if two successive transversal sections have not the same formulation, a mixed link is generated for the following combinations: 1D/3D and 1D/2D. The 2D/3D link is not managed by the MAI modulus.

Conditions of use

- Must be the first command of the file
- The default format is .msh.
- The option FICHIER UNIQUE is the default option.

Methodological advice

- The choice of the format triggers the method to named mesh groups of the model (within the meaning of Code_Aster). With the MED format, they are named according to the following principle: Vou+No. segment. For example, the No. 15 segment is assigned with the string “Vou15”. With the MSH format, strings are not allowed in the name of mesh groups, so they are numbered as follows: 100000+No. segment*1000. The user can call these mesh groups in the command file of Code_Aster by using the prefix ‘GM’. For example, the 24th segment is named “GM124000”. Furthermore, a mesh group is also created for every surface of a segment: in both cases, it is referenced as follows: “GM”+”100000+No. segment*1000+No. of the surface of the segment”. For example, the associate mesh group of the 3rd elementary surface of the 7th segment can be called: “GM107003”.

- The finite element modulus generates a geometry file, a mesh file and a command file for a finite element software. At the moment, a command file for Code_aster is created with the option FICHIER UNIQUE for volume elements, surface elements and line elements. The command file includes the reading of the mesh, the assignment of the model, the material definition and their assignment, the mesh characteristics, the definition of the prestressed forces and the dead weight and the linear static calculus of the prestressing and the dead weight. The user will have to define the boundary conditions and could conduct the study by setting other loads and the calculations relative to these loads.

Examples

```
DISCRETISATION FORMAT MSH FICHIER UNIQUE;
..........
```

Linked Commands

FIN;

13.2 - STRUCTURE

STRUCTURE nom_structure ;

Parameters

- nom_structure : name of a saved structure.

Functions

This command recalls a saved structure, for which a geometry file, a mesh file and a command file for a finite element computation software are created.

Conditions of use

- ☐ The called structure must already be recorded in the database, using the SAUVER command in the PH3 module (see Chapter 8).
- ☐ The saved structure must not provoke any instabilities (for example, due to the non-activation of articulations linking the parts of the structure).
- ☐ A saved structure must be called before using the LOGICIEL, MAILLAGE POUTRE and MAILLAGE ELEMENT commands which act on it, and its effect is remanent.
- ☐ This command can't be repeated so several STRUCTURES can't be replaced in a single file.

Examples

```
DISCRETISATION FORMAT MSH FICHIER UNIQUE;

STRUCTURE STRUTEST;

LOGICIEL ASTER

MODELISATION '3D';

MAILLAGE POUTRE VOLU
```

Linked commands

MAILLAGE POUTRE ; MAILLAGE ELEMENT

13.3 - SOFTWARE

LOGICIEL logiciel_cible

[MODELISATION modelisation_globale];

Parameters

logiciel_cible : finite elements software for which the MAI module will generate a command file. For now : ASTER

- modelisation_globale : finite elements type used in the model

Functions

This command specifies the target software which will be used to use the command file generated by the MAI module. For now, only the Code_Aster software (http://www.code-aster.org/) is available through the MAI module.

With this command, the user can specify the type of the finite elements.

Conditions of use

- ☐ If this command appears, it must immediately follow the STRUCTURE command.
- ☐ When the Code_Aster software is used (by default), the different values that can be entered in the parameter modelisation_globale are described in the documentation of the AFFE_MODELE command of Code_Aster, whose reference ic U4.41.01. Besides, the use of this parameter, even though it is optional, is recommended for a use od the command file with Code_Aster. Indeed, it is recommended to assign a model to all elements, even if some elements will be reassign if the model has several types.

Examples

```
MAILLAGE FORMAT MED;

LOGICIEL ASTER
MODELISATION 'POU_D_T';
..........
```

Linked commands

LOGICIEL ; MAILLAGE POUTRE VOLUMIQUE ; MAILLAGE POUTRE SURFACIQUE ; MAILLAGE POUTRE LINEIQUE ; MAILLAGE ELEMENTS ;

13.4 – MESH VOLUME BEAM

MAILLAGE POUTRE VOLUMIQUE

$$* \left\{ \begin{array}{l} \text{TRONCON nu\_deb nu\_fin} \\ \text{ALGORITHME} \left\{ \begin{array}{l} \text{DEL3D} \\ \text{FRONT3D} \\ \text{MMG3D} \end{array} \right\} \\ \text{LISSAGE nb\_liss} \\ \text{ORDRE nu\_ord} \\ \text{ECHELLE echelle} \\ \text{MINI t\_mini} \\ \text{MAXI t\_maxi} \\ \text{COURBE} \\ \text{OPTIMISER} \\ \text{PERTURBATION pert} \\ \text{VERIFIER} \\ \text{MODELISATION modelisation} \\ \text{PRECONTRAINTE} \\ \\ \\ \end{array} \right\}$$

l_pout;

Parameters

- nu_deb, nu_fin: numbers of points of the fiber reference which indicate the section of the l_pout beam(s) to which the parameters will be applied.
- nb_liss: number of steps in the smoothing of the mesh.
- nu_ord: order of the generated elements
- ech: characteristic length of the meshes.
- t_mini: minimum size of the meshes
- t_maxi: maximum size of the meshes
- pert: random disturbance factor
- modelisation: model of the section elements, as it is defined in the AFFE_MODELE command of the software Code_Aster (U4.41.01).
- l_pout: list of the beam numbers for which the mesh parameters are set.

Functions

This command allows generating a volume finite element model of a part of one or several beams.

The OPTIMISER option allows improving the tetrahedrons quality.

The COURBE option allows calculating the mesh size from the curvature.

The VERIFIER option allows verifying the homogeneity of the mesh.

The PRECONTRAINTE option allows taking the prestressing into account: in that case, the linear meshes of the prestressing cables will be created and the command file of the finite element study will include the model of the cables and the prestressing forces.

Conditions of use

- The volume finite elements model is possible only if the beam has been defined with the commands SECTION MASSIVE and SECTION CONTOURS ENTIERE.
- If the beam is not uniform in terms of section topology on its whole length, the mesh will be "cut" at its discontinuity with the FICHIER MULTIPLES method, or the link will be forced with the FICHIER UNIQUE option.
- The elements order must be strictly positive and lower or equal to 5.
- For the volume models, the DEL3D algorithm is the fastest and the strongest. It is the default option.
- If the TRONCON option is not set, the parameters applies on all the beams of l_pout.
- The MAI module automatically create mesh groups (within the meaning of Code_Aster, defined as Physical in the geometry GMSH file). A mesh group is created for every segment of the volume part. With the MED format, they are named according to the following principle: Vou+No. segment. For example, the No. 15 segment is assigned with the string "Vou15". With the MSH format, strings are not allowed in the name of mesh groups, so they are numbered as follows: 100000+No. segment*1000. The user can call these mesh groups in the command file of Code_Aster by using the prefix 'GM'. For example, the 24th segment is named "GM124000". Furthermore, a mesh group is also created for every transverse section: in both cases, it is referenced as follows: "GM"+"100000+No. reference fiber point*1000+1". For example, the associate mesh group of the section of the 12th point of the reference fiber can be called: "GM112001".
- If the PRECONTRAINTE option is set even, when there is no cable in the called part, a warning message will appear without interrupting the calculation.
- The "finite element" module create a command file for a finite element software. For the moment, a command file for Code_Aster is correctly created in the case of a single file. The command file includes the reading of the mesh, the model assignment, the definition of the materials and their assignments, the meshes characteristics, the setting of the prestressing forces and of the dead weight and the linear calculation of these effects on the structure. The user will have to set the boundary conditions and can extend the study by adding the other loads and the calculation of their effects.

Examples

```
MAILLAGE POUTRE VOLUMIQUE
TRONCON 5 27
ALGORITHME DEL3D
OPTIMISER
ORDRE 1
MINI 0.3 MAXI 0.8
1;
```

Linked commands

MAILLAGE POUTRE SURFACIQUE ; MAILLAGE POUTRE LINEIQUE ; MAILLAGE ELEMENTS;

13.5 - MESH SURFACE BEAM

MAILLAGE POUTRE SURFACIQUE

$$* \left\{ \begin{array}{l} \text{TRONCON nu\_deb nu\_fin} \\ \text{ALGORITHME} \left\{ \begin{array}{l} \text{DEL2D} \\ \text{FRONT2D} \\ \text{MESHADAPT} \end{array} \right\} \\ \text{LISSAGE nb\_liss} \\ \text{ORDRE nu\_ord} \\ \text{ECHELLE echelle} \\ \text{MINI t\_mini} \\ \text{MAXI t\_maxi} \\ \text{COURBE} \\ \text{OPTIMISER} \\ \text{PERTURBATION pert} \\ \text{VERIFIER} \\ \text{RECOMBINER discretisation} \\ \text{MODELISATION modelisation} \\ \text{PRECONTRAINTE} \end{array} \right\}$$

l_pout;

Parameters

- nu_deb, nu_fin: numbers of points of the fiber reference which indicate the section of the l_pout beam(s) to which the parameters will be applied.
- nb_liss: number of steps in the smoothing of the mesh.
- nu_ord: order of the generated elements
- ech: characteristic length of the meshes.
- t_mini: minimum size of the meshes
- t_maxi: maximum size of the meshes
- pert: random disturbance factor
- discretisation: number of generated points on the lines of the whole part of the beam during the generation of the mesh.
- modelisation: model of the section elements, as it is defined in the AFFE_MODELE command of the software Code_Aster (U4.41.01).
- l_pout: list of the beam numbers for which the mesh parameters are set.

Functions

This command allows generating a surface finite element model of a part of one or several beams.

The OPTIMISER option allows improving the tetrahedrons quality.

The COURBE option allows calculating the mesh size from the curvature.

The VERIFIER option allows verifying the homogeneity of the mesh.

The RECOMBINER option transforms a mesh consisting only of triangles into a mesh mixing triangles and quadrangles.

The PRECONTRAINTE option allows taking the prestressing into account: in that case, the linear meshes of the prestressing cables will be created and the command file of the finite element study will include the model of the cables and the prestressing forces.

Conditions of use

- The surface finite elements model is possible only if the beam has been defined with the commands SECTION PAROIS and SECTION CONTOURS ENTIERE.
- If the beam is not uniform in terms of section topology on its whole length, the mesh will be "cut" at its discontinuity with the FICHIER MULTIPLES method, or the link will be forced with the FICHIER UNIQUE option.
- The elements order must be strictly positive and lower or equal to 5.
- For the plate models, the MESHADAPT algorithm is adapted to the complex curved surfaces. The FRONT2D algorithm provides the best quality of the mesh. The DEL2D algorithm, which is the default algorithm, is the fastest for big surfaces.
- If the TRONCON option is not set, the parameters applies on all the beams of l_pout.
- The MAI module automatically create mesh groups (within the meaning of Code_Aster, defined as Physical in the geometry GMSH file). A mesh group is created for every segment of the surface part. With the MED format, they are named according to the following principle: Vou+No. segment. For example, the No. 15 segment is assigned with the string "Vou15". With the MSH format, strings are not allowed in the name of mesh groups, so they are numbered as follows: 100000+No. segment*1000. The user can call these mesh groups in the command file of Code_Aster by using the prefix 'GM'. For example, the 24th segment is named "GM124000". Furthermore, a mesh group is also created for every surface of a segment: in both cases, it is referenced as follows: "GM"+"100000+No. segment*1000+No. of the surface of the segment". For example, the associate mesh group of the 3rd elementary surface of the 7th segment can be called: "GM107003".
- If the PRECONTRAINTE option is set even, when there is no cable in the called part, a warning message will appear without interrupting the calculation.
- The "finite element" module create a command file for a finite element software. For the moment, a command file for Code_Aster is correctly created in the case of a single file. The command file includes the reading of the mesh, the model assignment, the definition of the materials and their assignments, the meshes characteristics, the setting of the prestressing forces and of the dead weight and the linear calculation of these effects on the structure. The user will have to set the boundary conditions and can extend the study by adding the other loads and the calculation of their effects.

Examples

```
MAILLAGE POUTRE SURFACIQUE
TRONCON 12 32
ALGORITHME MESHADAPT
```

```
OPTIMISER
MAXI 0.8
VERIFIER
MODELISATION 'DKT'
1 3;
```

Linked commands

MAILLAGE POUTRE VOLUMIQUE ; MAILLAGE POUTRE LINEIQUE ;

MAILLAGE ELEMENTS;

13.6 – MESH LINEAR BEAM

MAILLAGE POUTRE LINEIQUE

$$* \left\{ \begin{array}{l} \text{TRONCON nu\_deb} \quad \text{nu\_fin} \\ \text{MODELISATION modelisation} \\ \text{PRECONTRAINTE} \end{array} \right\}$$

l_pout;

Parameters

- nu_deb, nu_fin: numbers of points of the fiber reference which indicate the section of the l_pout beam(s) to which the parameters will be applied.
- modelisation: model of the section elements, as it is defined in the AFFE_MODELE command of the software Code_Aster (U4.41.01).
- l_pout: list of the beam numbers for which the mesh parameters are set.

Functions

This command allows generating a linear finite element model of a part of one or several beams.

The PRECONTRAINTE option allows taking the prestressing into account: in that case, the linear meshes of the prestressing cables will be created and the command file of the finite element study will include the model of the cables and the prestressing forces.

Conditions of use

- The linear finite elements model is possible for any modeling method to define the model in PCP.
- If the TRONCON option is not set, the parameters applies on all the beams of l_pout.
- The MAI module automatically create mesh groups (within the meaning of Code_Aster, defined as Physical in the geometry GMSH file). A mesh group is created for every segment of the volume part. With the MED format, they are named according to the following principle: Vou+No. segment. For example, the No. 15 segment is assigned with the string “Vou15”. With the MSH format, strings are not allowed in the name of mesh groups, so they are numbered as follows: 100000+No. segment*1000. The user can call these mesh groups in the command file of Code_Aster by using the prefix ‘GM’. For example, the 24th segment is named “GM124000”.
- If the PRECONTRAINTE option is set even, when there is no cable in the called part, a warning message will appear without interrupting the calculation.

- The “finite element” module create a command file for a finite element software. For the moment, a command file for Code_Aster is correctly created in the case of a single file. The command file includes the reading of the mesh, the model assignment, the definition of the materials and their assignments, the meshes characteristics, the setting of the prestressing forces and of the dead weight and the linear calculation of these effects on the structure. The user will have to set the boundary conditions and can extend the study by adding the other loads and the calculation of their effects.

Examples

```
MAILLAGE POUTRE LINEIQUE
TRONCON 12 32
MODELISATION 'POU_D_T'
1;
```

Linked commands

MAILLAGE POUTRE VOLUMIQUE ; MAILLAGE POUTRE SURFACIQUE ; MAILLAGE ELEMENTS;

13.7 – MESH ELEMENTS

$$\text{MAILLAGE ELEMENTS}\quad\left\{\begin{array}{l}\text{ARTICULATIONS}\\\text{BIARTICULES}\\\text{STANDARDS}\\\text{RIGIDES}\end{array}\right\};$$

Functions

This command generates the linear finite element model of the non-beam elements.

Conditions of use

- It is necessary to repeat the command for each type of element.
- With the FICHIER UNIQUE option, these elements are added to the global geometry file. Otherwise, they are created in separate files depending on their type.

Examples

```
MAILLAGE ELEMENTS RIGIDES;
..........
MAILLAGE ELEMENTS STANDARDS;
..........
```

Linked commands

MAILLAGE POUTRE VOLUMIQUE ; MAILLAGE POUTRE SURFACIQUE ; MAILLAGE POUTRE LINEIQUE;

13.8 - END

FIN ;

Functions

This command marks the end of the execution of the MAI module; all following commands are ignored.

Examples

```
MAILLAGE FORMAT MSH;
...............
FIN;
```

Annex A

Reference Documents

Name	More complete description	Publication ref.
IP 2	Circular No. 73153 of August 13, 1973, on the use of prestressed concrete [...]	BO 7364a Special Issue
Issue 61	CPC Issue 61 Title II DESIGN, CALCULATION AND TESTING OF ENGINEERING STRUCTURES - Loading programs and testing of road bridges	BO 7221a (1981 reprint)
Circular 83	Circular letter of July 20, 1983 EXCEPTIONAL LOADS - STANDARD CONVOY DEFINITION AND RULES FOR TESTING OF ENGINEERING STRUCTURES	DR-R/EG3
CEB Model Code	EURO INTERNATIONAL CONCRETE COMMITTEE CEB-FIP MODEL CODE - DESIGN CODE	T. Telford 1990
CEB 90	EURO INTERNATIONAL CONCRETE COMMITTEE CEB FIP 1990 MODEL CODE DESIGN CODE	T. Telford 1993
BAEL 83	CCTG Issue 62 Title I Section I Technical design rules and calculation for reinforced concrete structures, according to the limit states method	BO 83-45a Special Issue
BPEL 91 (Revised 99)	CCTG Issue 62 Title I Section II Technical design rules and calculation for prestressed concrete structures, according to the limit states method	BO 999 Special Issue
Eurocode 1	Eurocode 1 -Part 3 Basis of calculation and actions on structures - Loads on bridges due to traffic and DAN (IC: P06-103)	AFNOR XP ENV 1991-3 (October 1997)
Eurocode 2	Eurocode 2 -parts 1-x and 2 Concrete structure calculations (IC: P18-7xx)	AFNOR ENV 1992-1-x and 1992-2
Eurocode 8	Eurocode 8 -part 1-1 Design and dimensioning of structures for earthquake resistance - General rules, seismic action and general requirements for structures and DAN (ICS: 91.120.20)	AFNOR XP ENV 1998-1-1
AFPS 92	French Association for Earthquake Engineering - AFPS 92 Guide for seismic protection of bridges	ENPC press
CDS	CDS PROGRAM (Calculation Of Sections) REFERENCE MANUAL (Version 3.4)	SETRA (May 1991)
S606	CAN / CSA-S6-06 Canadian Highway Bridge Design Code	CSA Group publications
Doctoral thesis	Contribution to the non-linear geometric and material analysis of space frames in civil engineering. Application to engineering structures	F. Robert INSA Lyon (1999, 250 p.)

Table A.1 - List of Documents

Annex B

Rheological behavior of materials

Annex A has the full wording of the documents referenced in this annex under the condensed names "IP 2", "CEB Model Code", "CEB90", "BPEL 83", "BPEL 91", "S606", "AFREM 96" and "Eurocode 2".

For the J. Courbon creep law, see the article by the same author, published in issue 242 of the Annals of ITBTP (February 1968), entitled "THE INFLUENCE OF LINEAR CREEP ON THE EQUILIBRIUM OF STATICALLY INDETERMINATE PRESTRESSED CONCRETE SYSTEMS ".

The numbers of rheological behavior laws of base materials (loi_fluage, loi_module and loi_retrait), the reference Young's modulus (module_reference), and extra parameters (para_supp$_1$, para_supp$_2$ and para_supp$_3$) relate to the MATERIAU command of the GE1 (see Chapter 3) or PH1 module (see Chapter 6).

The values to be introduced must be taken from one of the tables below.

For the formulation and notation, refer to the references cited in the table headings.

Parameters	Description
loi_fluage	0: material with no creep 1: material creep according to the J. Courbon law
loi_module	0: the instantaneous Young's modulus is constant and equal to the reference modulus
loi_retrait	0: material with no shrinkage
$para\_supp_1$	instantaneous Young's modulus at 28 days
$para\_supp_2$	not used
$para\_supp_3$	not used

Table B.1 - Basic Laws

Parameters	Description
loi_fluage	2: the material creeps according to the formulation law
loi_module	1: the instantaneous Young's modulus is constant and equal to the 28-day modulus 2: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	1: the material shrinks according to the formulation law
$para\_supp_1$	instantaneous Young's modulus at 28 days
$para\_supp_2$	formulation parameter k_b; $0.6 \leq k_b \leq 2.0$
$para\_supp_3$	formulation parameter k_p; $0.5 \leq k_p \leq 1.0$

Table B.2 - IP 2 laws for conventional concrete

Parameters	Description
loi_fluage	6: the material creeps according to the formulation law
loi_module	9: the instantaneous Young's modulus is constant and equal to the 28-day modulus 10: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	5: the material shrinks according to the formulation law
$para\_supp_1$	instantaneous Young's modulus at 28 days
$para\_supp_2$	formulation parameter k_b; $0.6 \leq k_b \leq 2.0$
$para\_supp_3$	formulation parameter k_p; $0.5 \leq k_p \leq 1.0$

Table B.3 - IP 2 laws for lightweight concrete

Parameters	Description
loi_fluage	4: the material creeps according to the formulation law
loi_module	5: the instantaneous Young's modulus is constant and equal to the 28-day modulus 6: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	3: the material shrinks according to the formulation law
$para\_supp_1$	instantaneous Young's modulus at 28 days
$para\_supp_2$	formulation parameter α; $0.001 \leq \alpha \leq 5.0$
$para\_supp_3$	formulation parameter k_1

Table B.4 - CEB Model Code laws

Parameters	Description
loi_fluage	3: the material creeps according to the formulation law
loi_module	3: the instantaneous Young's modulus is constant and equal to the 28-day modulus 4: the instantaneous Young's modulus varies according to the BPEL 83 formulation law 14: the instantaneous Young's modulus varies according to the BPEL 91 formulation law, for concrete having a compressive strength f_{cj} less than or equal to 40 Mpa 16: same as above, when f_{cj} is greater than 40 Mpa
loi_retrait	2: the material shrinks according to the formulation law
para_supp$_1$	instantaneous Young's modulus at 28 days
para_supp$_2$	unused parameter
para_supp$_3$	formulation parameter k_s; $0.5 \le k_s \le 1.0$

Table B.5 - BPEL laws for conventional concrete

Parameters	Description
loi_fluage	5: the material creeps according to the formulation law
loi_module	7: the instantaneous Young's modulus is constant and equal to the 28-day modulus 8: the instantaneous Young's modulus varies according to the BPEL 83 formulation law
loi_retrait	4: the material shrinks according to the formulation law
para_supp$_1$	instantaneous Young's modulus at 28 days
para_supp$_2$	formulation parameter E_a
para_supp$_3$	formulation parameter k_s; $0.5 \le k_s \le 1.0$

Table B.6 - BPEL laws for lightweight concrete

Parameters	Description
loi_fluage	10: the material creeps according to the formulation law
loi_module	3: the instantaneous Young's modulus is constant and equal to the 28-day modulus 4: the instantaneous Young's modulus varies according to the BPEL 83 formulation law 14: the instantaneous Young's modulus varies according to the BPEL 91 formulation law, for concrete having a compressive strength, f_{cj}, less than or equal to 40 Mpa 16: same as above, when f_{cj} is greater than 40 Mpa
loi_retrait	9: the material shrinks according to the formulation law
para_supp$_1$	instantaneous Young's modulus at 28 days
para_supp$_2$	indicates the presence of silica fume, if negative, or absent, if it is positive or zero
para_supp$_3$	formulation parameter k_s; $0.5 \le k_s \le 1.0$

Table B.7 - BPEL Laws (99, Annex 14 for HP concrete)

Parameters	**Description**
loi_fluage	9: the material creeps according to the formulation law
loi_module	17: the instantaneous Young's modulus is constant and equal to the 28-day modulus 18: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	8: the material shrinks according to the formulation law
$para\_supp_1$	instantaneous Young's modulus at 28 days
$para\_supp_2$	indicates the setting rate of the cement: 1.0 fast, 2.0 standard, 3.0 slow
$para\_supp_3$	coefficient to account for the presence of passive steel bars, according to the same principle as the BPEL k_s.

Table B.8 - Eurocode 2 laws - ENV Version

Parameters	**Description**
loi_fluage	11: the material creeps according to the formulation law
loi_module	19: the instantaneous Young's modulus is constant and equal to the 28-day modulus 20: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	11: the material shrinks according to the formulation law
$para\_supp_1$	instantaneous Young's modulus at 28 days: Ecm value
$para\_supp_2$	Its absolute value indicates the strength class: 1.0 class R, 2.0 class N, 3.0 class S; its sign indicates whether creep and shrinkage deformation are weighted (negative) or not (positive) by the coefficient γlt of section B.105 of EN 1992-2:2005.
$para\_supp_3$	coefficient to account for the presence of passive steel bars, according to the same principle as the BPEL k_s.

Table B.9 - Eurocode 2 laws - EN 1992-1-1: 2004

Parameters	**Description**
loi_fluage	12: the material creeps according to the formulation law
loi_module	19: the instantaneous Young's modulus is constant and equal to the 28-day modulus 20: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	12: the material shrinks according to the formulation law
$para\_supp_1$	instantaneous Young's modulus at 28 days: Ecm value
$para\_supp_2$	Its absolute value indicates the strength class: 1.0 class R, 2.0 class N, 3.0 class S; its sign indicates whether creep and shrinkage deformation are weighted (negative) or not (positive) by the coefficient γlt of section B.105 of EN 1992-2:2005.
$para\_supp_3$	coefficient to account for the presence of passive steel bars, according to the same principle as the BPEL k_s.

Table B.10 - Eurocode 2 laws - BHP without silica fume - EN 1992-2: 2005 – Annex B.103

Parameters	Description
loi_fluage	13: the material creeps according to the formulation law
loi_module	19: the instantaneous Young's modulus is constant and equal to the 28-day modulus 20: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	13: the material shrinks according to the formulation law
para_supp$_1$	instantaneous Young's modulus at 28 days: Ecm value
para_supp$_2$	Its absolute value indicates the strength class: 1.0 class R, 2.0 class N, 3.0 class S; its sign indicates whether creep and shrinkage deformation are weighted (negative) or not (positive) by the coefficient γlt of section B.105 of EN 1992-2:2005.
para_supp$_3$	coefficient to account for the presence of passive steel bars, according to the same principle as the BPEL k_s.

Table B.11 - Eurocode 2 laws - BHP with silica fume – EN 1992-2: 2005 – Annex B.103

Parameters	Description
loi_fluage	14: the material creeps according to the formulation law
loi_module	21: the instantaneous Young's modulus is constant and equal to the 28-day modulus 22: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	106: the material shrinks according to the formulation law
para_supp$_1$	instantaneous Young's modulus at 28 days: Eci value
para_supp$_2$	Its absolute value indicates the strength class: 1.0 class RS, 2.0 class N and R, 3.0 class SL
para_supp$_3$	coefficient to account for the presence of passive steel bars, according to the same principle as the BPEL k_s.

Table B.12 - CEB 1990 Model Code laws

Parameters	Description
loi_fluage	15: the material creeps according to the formulation law
loi_module	23: the instantaneous Young's modulus is constant and equal to the 28-day modulus 24: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	107: the material shrinks according to the formulation law
para_supp$_1$	instantaneous Young's modulus at 28 days: Ec,28 value
para_supp$_2$	Cement type: 1 for a type 10 cement normal cure, 2 for a type 30 cement normal cure, 3 for a type 10 cement accelerated cure, 4 for a type 30 cement accelerated cure
para_supp$_3$	coefficient to account for the presence of passive steel bars, according to the same principle as the BPEL k_s.

Table B.13 - Canadian Code CSA-S6-06 laws

Parameters	Description
loi_fluage	16: the material creeps according to the formulation law
loi_module	19: the instantaneous Young's modulus is constant and equal to the 28-day modulus 20: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	16: the material shrinks according to the formulation law
para_supp$_1$	instantaneous Young's modulus at 28 days: Ecm value
para_supp$_2$	Its absolute value indicates the strength class: 1.0 class R, 2.0 class N, 3.0 class S; its sign indicates whether creep and shrinkage deformation are weighted (negative) or not (positive) by the coefficient γlt of section B.105 of EN 1992-2:2005.
para_supp$_3$	coefficient to account for the presence of passive steel bars, according to the same principle as the BPEL k_s.
para_supp$_4$	coefficient k_{rd} to account for the deformation due to drying shrinkage : $\varepsilon_{cs}=k_{rd}\varepsilon_{cd}+k_{re}\varepsilon_{ca}$; by default, $k_{rd} = 1.0$ (equation 3.8)
para_supp$_5$	coefficient k_{re} to account for the deformation due to endogenous shrinkage: $\varepsilon_{cs}=k_{rd}\varepsilon_{cd}+k_{re}\varepsilon_{ca}$; by default, $k_{re} = 1.0$ (equation 3.8)
para_supp$_6$	coefficient k_{fd} to account for the conventional deferred deformation due to drying creep : $\varepsilon_{cc}(t,t_0)=(\sigma(t_0)/E_c)(k_{fe}\varphi_b(t,t_0)+k_{fd}\varphi_d(t,t_0))$; by default, $k_{fd} = 1.0$ (equation B.117)
para_supp$_7$	coefficient k_{fe} to account for the conventional deferred deformation due to own creep: $\varepsilon_{cc}(t,t_0)=(\sigma(t_0)/E_c)(k_{fe}\varphi_b(t,t_0)+k_{fd}\varphi_d(t,t_0))$; by default, $k_{fe} = 1.0$ (equation B.117)
para_supp$_8$	coefficient of creep speed k_{fc} modifying the law of creep evolution: $\beta_{bc}=0.4k_{fc}\exp(3.1f_{cm}(t_0)/f_{ck})$; by default, $k_{fc} = 1.0$ (équation B.120)

Table B.14 - Eurocode 2 laws - BHP without silica fume with advanced coefficients– EN 1992-2: 2005 – Annex B.103

Parameters	**Description**
loi_fluage	17: the material creeps according to the formulation law
loi_module	19: the instantaneous Young's modulus is constant and equal to the 28-day modulus 20: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	17: the material shrinks according to the formulation law
para_supp$_1$	instantaneous Young's modulus at 28 days: Ecm value
para_supp$_2$	Its absolute value indicates the strength class: 1.0 class R, 2.0 class N, 3.0 class S; its sign indicates whether creep and shrinkage deformation are weighted (negative) or not (positive) by the coefficient γlt of section B.105 of EN 1992-2:2005.
para_supp$_3$	coefficient to account for the presence of passive steel bars, according to the same principle as the BPEL k_s.
para_supp$_4$	coefficient k_{rd} to account for the deformation due to drying shrinkage : $\varepsilon_{cs}=k_{rd}\varepsilon_{cd}+k_{re}\varepsilon_{ca}$; by default, k_{rd} = 1.0 (equation 3.8)
para_supp$_5$	coefficient k_{re} to account for the deformation due to endogenous shrinkage: $\varepsilon_{cs}=k_{rd}\varepsilon_{cd}+k_{re}\varepsilon_{ca}$; by default, k_{re} = 1.0 (equation 3.8)
para_supp$_6$	coefficient k_{fd} to account for the conventional deferred deformation due to drying creep : $\varepsilon_{cc}(t,t_0)=(\sigma(t_0)/E_c)(k_{fe}\varphi_b(t,t_0)+k_{fd}\varphi_d(t,t_0))$; by default, k_{fd} = 1.0 (equation B.117)
para_supp$_7$	coefficient k_{fe} to account for the conventional deferred deformation due to own creep: $\varepsilon_{cc}(t,t_0)=(\sigma(t_0)/E_c)(k_{fe}\varphi_b(t,t_0)+k_{fd}\varphi_d(t,t_0))$; by default, k_{fe} = 1.0 (equation B.117)
para_supp$_8$	coefficient of creep speed k_{fc} modifying the law of creep evolution: $\beta_{bc}=0.37k_{fc}\exp(2.8f_{cm}(t_0)/f_{ck})$; by default, k_{fc} = 1.0 (équation B.120)

Table B.15 - Eurocode 2 laws - BHP with silica fume with advanced coefficients– EN 1992-2: 2005 – Annex B.103

Parameters	Description
loi_fluage	18: the material creeps according to the formulation law
loi_module	19: the instantaneous Young's modulus is constant and equal to the 28-day modulus 20: the instantaneous Young's modulus varies according to the formulation law
loi_retrait	18: the material shrinks according to the formulation law
para_supp$_1$	instantaneous Young's modulus at 28 days: Ecm value
para_supp$_2$	Its absolute value indicates the strength class: 1.0 class R, 2.0 class N, 3.0 class S; its sign indicates whether creep and shrinkage deformation are weighted (negative) or not (positive) by the coefficient γlt of section B.105 of EN 1992-2:2005.
para_supp$_3$	coefficient to account for the presence of passive steel bars, according to the same principle as the BPEL k_s.
para_supp$_4$	coefficient k_{rd} to account for the deformation due to drying shrinkage : $\varepsilon_{cs}=k_{rd}\varepsilon_{cd}+k_{re}\varepsilon_{ca}$; by default, k_{rd} = 1.0 (equation 3.8)
para_supp$_5$	coefficient k_{re} to account for the deformation due to endogenous shrinkage: $\varepsilon_{cs}=k_{rd}\varepsilon_{cd}+k_{re}\varepsilon_{ca}$; by default, k_{re} = 1.0 (equation 3.8)
para_supp$_6$	coefficient k_{cr} affecting drying shrinkage evolution: $\beta_{ds}(t,t_s)=(t-t_s)/((t-t_s)+0,04k_{cr}h^{3/2}$; by default, k_{cr}=1 (equation3.10)
para_supp$_7$	coefficient k_{fd} to account for the conventional deferred deformation due to creep: $\varepsilon_{cc}(\infty,t_0)=\varphi(\infty,t_0).(\sigma_c/E_c)$; by default, k_f = 1.0 (equation 3.6)
para_supp$_8$	coefficient of creep speed k_{fc} modifying the law of creep evolution : $\beta_c(t,t_0)=((t-t_0)/(\beta_H+t-t_0))^{kfc}$; by default, k_{fc} = 0.3 (équation B.7)

Table B.16 - Eurocode 2 laws with advanced coefficients– EN 1992-1-1: 2004

Annex C

Traffic examples

This annex contains a guidance skeleton, to establish data for the TRAFFIC sub-module, which uses its commands and their options exhaustively, and predefined traffic corresponding to the options TRAFFIC AL, BCP, C1P and E_M1C1_C of the ENV module.

C.1 - TRAFFIC SKELETON

```
DEBUT  T_XXXXXX;

$ title of the TRAFIC in the form of comments
$ ----------------------------------------------------------------------
$ the most frequently used parameter values are
$ (parameters that are not mentioned have at least two frequent values)

$ for BANDE_CENTRALE
$    band type (band_xxx)          ==> (band_n_char)

$ for ROUE_VEHICULE
$    wheel type (wheel_xxxx)         ==> (wheel_cons)
$    length type (length_xxxx)       ==> (length_impo)
$    min lg                          ==> 0.00
$    max imposed lg                  ==> 1.00
$    min imposed lg                  ==> 1.00
$    wheel_weighting f               ==> 1.00
$    n°_val                          ==> 0

$ for ESSIEU_VEHICULE
$    axle_f_weighting                ==> 1.00

$ for VEHICULE
$    vehicle_f_weighting             ==> 0.00
$    unif removal type (xxxx_unif)   ==> (leave_unif)

$ for CONVOI_VEHICULES
$    convoy type (convoy_inde_xxxx)  ==> (convoy_inde_1)

$ for CONCENTREE
$    type_of concentrated(vehicle_xxxx) ==> (vehicle_droi)
$    f_conc weighting                   ==> 0.00

$ for DENSITE_UNIFORME
$    type_of density (density_xxxx)  ==> (densite_cons)
$    min lg                          ==> 0.00
$    max imposed lg                  ==> 1.00
$    min_imposed lg                  ==> 1.00
$    n°_val                          ==> 0

$ for UNIFORME
$    type_of uniform (arees_xxxx)    ==> (cumu areas)
$    type_of length (length_xxxx)    ==> (calc length)
$    f_unif weighting                ==> 0.00

$ for CHARGE_VOIE
$    type_of load (load_xxxx)        ==> (gene load)
$    unif removal type (xxxx_unif)   ==> (leave_unif)
$    f_unif weighting                ==> 0.00
$ ----------------------------------------------------------------------

$ definition of the loadable length, case 1
LONGUEUR_CHARGEABLE  (longueur_1) SUPPORT;

$ definition of the loadable length, case 2
<x_zone_inf = >
<x_zone_sup = >
LONGUEUR_CHARGEABLE  (longueur_1, x_zone_inf, x_zone_sup);
```

```
$ definition of the loadable width, case 1
LARGEUR_CHARGEABLE  (largeur_1) CIRCULATION CHAUSSEES;

$ definition of the loadable width, case 2
LARGEUR_CHARGEABLE  (largeur_1) CIRCULATION TROTTOIRS;

$ definition of the loadable width, case 3
<y_zone_inf = >
<y_zone_sup = >
LARGEUR_CHARGEABLE  (largeur_1, y_zone_inf, y_zone_sup);

$ definition of the central reservation, case 1
BANDE_CENTRALE  (bande_1, bande_xxxx) CIRCULATION;

$ definition of the central reservation, case 2
<y_bande_inf = >
<y_bande_sup = >
BANDE_CENTRALE  (bande_1, bande_xxxx, y_bande_inf, y_bande_sup);

$ definition of type i vehicle wheel
<impact_x       = >
<impact_y       = >
<lg_min         = >
<lg_imposee_max = >
<lg_imposee_min = >
<coef_a         = >
<ponder_f_roue  = >
<nb_val         = >
ROUE_VEHICULE  (roue_i, roue_xxxx, impact_x, impact_y, longueur_xxxx,
                lg_min, lg_imposee_max, lg_imposee_min, coef_a,
                ponder_f_roue, nb_val) [L1, Q1, L2, Q2, ..., ...];

$ definition of type i vehicle axle
<ponder_f_essi = >
<nb_roues      = >
ESSIEU_VEHICULE  (essieu_i, ponder_f_essi, nb_roues)
                 (roue_1, roue_2, ...) (y_roue1, y_roue2, ...);

$ definition of type i vehicle
<ponder_f_vehi = >
<dist_avant    = >
<dist_apres    = >
<enleve_avant  = >
<enleve_apres  = >
<nb_essieux    = >
VEHICULE  (vehicule_i, ponder_f_vehi, xxxx_unif, dist_avant,
           dist_apres, enleve_avant, enleve_apres, nb_essieux,
          (essieu_1, essieu_2, ...) (x_essieu1, x_essieu2, ...);

$ definition of convoy of type i vehicles
<nb_vehicules = >
CONVOI_VEHICULES  (convoi_i, convoi_inde_xxxx, pr_inutilise,
                   nb_vehicules) (vehicule_1, vehicule_2, ...);

$ definition of type i concentrated load
<ponder_d_conc = >
<ponder_f_conc = >
<coef_rech     = >
CONCENTREE  (charge_conc_i, vehicule_xxxx, convoi_i,
             ponder_d_conc, ponder_f_conc, coef_rech);
```

```
$ definition of the density of type i uniform load
<lg_min         = >
<coef_b         = >
<coef_c         = >
<coef_d         = >
<lg_imposee_max = >
<lg_imposee_min = >
<nb_val         = >
DENSITE_UNIFORME  (densite_i, densite_xxxx, lg_min, pr_inutilise,
                   coef_b, coef_c, coef_d,
                   lg_imposee_max, lg_imposee_min, nb_val)
                   [l1, q1, l2, q2, ..., ...];

$ definition of type i uniform load
<ponder_d_unif = >
<ponder_f_unif = >
UNIFORME  (charge_unif_i, aires_xxxx, densite_i, longueur_xxxx,
           ponder_d_unif, ponder_f_unif);

$ definition of type i load applicable to a platform
CHARGE_VOIE  (charge_i, charge_xxxx,
              charge_conc_i, charge_unif_i, xxxx_unif);

$ definition of platforms
$ --------------------

$ type of platforms 0
VOIES  (voies_1, type_voies_0, degres_0);

$ type 1 platforms, case 1
<nb_clas = >
<nb_degr = >
VOIES  (voies_1, type_voies_1, degres_xxxx)  CIRCULATION VOIES
       (nb_clas) (v0_1, v0_2, ...) (nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       .............................;

$ type 1 platforms, case 2
<v0      = >
<v       = >
<nb_clas = >
<nb_degr = >
VOIES  (voies_1, type_voies_1, degres_xxxx)  CIRCULATION CLASSE
       (v0, v, nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       .............................;

$ type 1 platforms, case 3
<v0      = >
<v       = >
<nb_degr = >
VOIES  (voies_1, type_voies_1, degres_xxxx, v0, v, nb_degr)
       (coef_degr1, coef_degr2, ...);

$ type 2 platforms, case 1
<nb_lcha = >
<nb_clas = >
<nb_degr = >
VOIES  (voies_1, type_voies_2, degres_xxxx)  CIRCULATION CLASSE
       (nb_lcha) (lcha_1, lcha_2, ...) (nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       .............................;
```

```
$ type 2 platforms, case 2
<nb_lcha = >
<nb_degr = >
VOIES  (voies_1, type_voies_2, degres_xxxx)
       (nb_lcha) (lcha_1, lcha_2, ...) (nb_degr)
       (coef_degr1, coef_degr2, ...);

$ type 3 platforms, case 1
<pas      = >
<nb_v_max = >
<lvoi_max = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_3, degres_xxxx)  CIRCULATION VOIES
       (pas, nb_v_max, lvoi_max, nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 3 platforms, case 2
<pas      = >
<nb_voies = >
<lvoi_max = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_3, degres_xxxx)  CIRCULATION LA_VOIES
       (pas, nb_voies, lvoi_max, nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 3 platforms, case 3
<pas      = >
<nb_v_max = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_3, degres_xxxx)  CIRCULATION NB_VOIES
       (pas, nb_v_max) (lvoi_1, lvoi_2, ...) (nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 3 platforms, case 4
<pas      = >
<nb_voies = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_3, degres_xxxx)  CIRCULATION CLASSE
       (pas, nb_voies) (lvoi_1, lvoi_2, ...) (nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 3 platforms, case 5
<pas      = >
<nb_voies = >
<nb_degr  = >
VOIES  (voies_1, type_voies_3, degres_xxxx)
       (pas, nb_voies) (lvoi_1, lvoi_2, ...) (nb_degr)
       (coef_degr1, coef_degr2, ...);
```

```
$ type 4 platforms, case 1
<pas      = >
<nb_voies = >
<lvoi_max = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_4, degres_xxxx)  CIRCULATION LA_VOIES
       (pas, nb_voies, lvoi_max, nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 4 platforms, case 2
<pas      = >
<nb_voies = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_4, degres_xxxx)  CIRCULATION CLASSE
       (pas, nb_voies) (lvoi_1, lvoi_2, ...) (nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 4 platforms, case 3
<pas      = >
<nb_voies = >
<nb_degr  = >
VOIES  (voies_1, type_voies_4, degres_xxxx)
       (pas, nb_voies) (lvoi_1, lvoi_2, ...) (nb_degr)
       (coef_degr1, coef_degr2, ...);

$ type 5 platforms, case 1
<nb_degr = >
VOIES  (voies_1, type_voies_5, degres_xxxx)  CIRCULATION TROTTOIRS
       (nb_degr)
       (coef_degr1, coef_degr2, ...);

$ type 5 platforms, case 2
<nb_clas = >
<nb_degr = >
VOIES  (voies_1, type_voies_5, degres_xxxx)  CIRCULATION CHAUSSEES
       (nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 5 platforms, case 3
<nb_clas = >
<nb_degr = >
VOIES  (voies_1, type_voies_5, degres_xxxx)  CIRCULATION VOIES
       (nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 5 platforms, case 4
<nb_voies = >
<nb_degr  = >
VOIES  (voies_1, type_voies_5, degres_xxxx, nb_voies)
       (yvoi_inf1, yvoi_sup1, yvoi_inf2, yvoi_sup2, ..., ...) (nb_degr)
       (coef_degr1, coef_degr2, ...);
```

```
$ type 6 platforms, case 1
<pas1     = >
<pas2     = >
<nb_v_max = >
<lvoi_max = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_6, degres_xxxx)  CIRCULATION VOIES
       (pas1, pas2, nb_v_max, lvoi_max, nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 6 platforms, case 2
<pas1     = >
<pas2     = >
<nb_voies = >
<lvoi_max = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_6, degres_xxxx)  CIRCULATION LA_VOIES
       (pas1, pas2, nb_voies, lvoi_max, nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 6 platforms, case 3
<pas1     = >
<pas2     = >
<nb_v_max = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_6, degres_xxxx)  CIRCULATION NB_VOIES
       (pas1, pas2, nb_v_max) (lvoi_1, lvoi_2, ...) (nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 6 platforms, case 4
<pas1     = >
<pas2     = >
<nb_voies = >
<nb_clas  = >
<nb_degr  = >
VOIES  (voies_1, type_voies_6, degres_xxxx)  CIRCULATION CLASSE
       (pas1, pas2, nb_voies) (lvoi_1, lvoi_2, ...) (nb_clas, nb_degr)
       (coef_degr1, coef_degr2, ...) $ class 1
       (coef_degr1, coef_degr2, ...) $ class 2
       ............................;

$ type 6 platforms, case 5
<pas1     = >
<pas2     = >
<nb_voies = >
<nb_degr  = >
VOIES  (voies_1, type_voies_6, degres_xxxx)
       (pas1, pas2, nb_voies) (lvoi_1, lvoi_2, ...) (nb_degr)
       (coef_degr1, coef_degr2, ...);

$ definition of the traffic loading conditions
MODALITES  (modalites_1, modalites_gene, largeur_1, longueur_1,
            no_inutilise, voies_1, bande_1);

$ definition of the traffic
<nb_charges = >
TRAFIC
TRAFIC  'INTITULE DU TRAFIC'
        (trafic_1, modalites_1, nb_charges) (charge_1, charge_2, ...);

FIN;
```

C.2 - LOAD A(l) OF ISSUE 61

```
DEBUT  T_AL;
$ Leaflet 61 - Title II of the C.P.C. - Load A(l)

$ definition of the loadable length
LONGUEUR_CHARGEABLE  (longueur_1) SUPPORT;

$ definition of the loadable width
LARGEUR_CHARGEABLE  (largeur_1) CIRCULATION CHAUSSEES;

$ definition of the central reservation
BANDE_CENTRALE  (bande_1, bande_n_char) CIRCULATION;

$ definition of the density of type 1 uniform load
<lg_min          = 0.00>
<coef_b          = 0.23>
<coef_c          = 36.00>
<coef_d          = 12.00>
<lg_imposee_max = 1.00>
<lg_imposee_min = 1.00>
<nb_val          = 0>
DENSITE_UNIFORME  (densite_1, densite_poly, lg_min, pr_inutilise,
                   coef_b, coef_c, coef_d,
                   lg_imposee_max, lg_imposee_min, nb_val);

$ definition type 1 uniform load
<ponder_d_unif = 1.00>
<ponder_f_unif = 1.00>
UNIFORME  (charge_unif_1, aires_comb, densite_1, longueur_calc,
           ponder_d_unif, ponder_f_unif);

$ definition of the load applicable to a type 1 platform
CHARGE_VOIE  (charge_1, charge_unif,
              no_inutilise, charge_unif_1, laissser_unif);

$ definition of the platforms
<nb_clas       = 3>
<nb_voies_max = 10>
<nb_degr       = (nb_voies_max)>
VOIES  (voies_1, type_voies_1, degres_g_glob)  CIRCULATION VOIES
       (nb_clas) 3.50 3.00 2.75 (nb_degr)
       1.00 1.00 0.90 0.75 6*0.70
       1.00                9*0.90
       0.90                9*0.80;

$ definition of the traffic load conditions
MODALITES  (modalites_1, modalites_gene, largeur_1, longueur_1,
            no_inutilise, voies_1, bande_1);

$ definition of the traffic
<nb_charges = 1>
TRAFIC
'FASCICULE 61 - TITRE II DU C.P.C. - CHARGE A(L)'
        (trafic_1, modalites_1, nb_charges, charge_1);

FIN;
```

C.3 - BC TRUCK OF ISSUE 61

```
DEBUT  T_BCP;
$ Leaflet 61 - Titre II of the C.P.C. - Bc load - positive direction

$ definition of the loadable length
LONGUEUR_CHARGEABLE  (longueur_1) SUPPORT;

$ definition of the loadable width
LARGEUR_CHARGEABLE  (largeur_1) CIRCULATION CHAUSSEES;

$ definition of the central reservation
BANDE_CENTRALE  (bande_1, bande_n_char) CIRCULATION;

$ definition of a type 1 vehicle wheel
<impact_x        = 0.00>
<impact_y        = 0.00>
<lg_min          = 0.00>
<lg_imposee_max = 1.00>
<lg_imposee_min = 1.00>
<coef_a          = 3.00>
<ponder_f_roue   = 1.00>
<nb_val          = 0>
ROUE_VEHICULE   (roue_1, roue_cons, impact_x, impact_y, longueur_impo,
                 lg_min, lg_imposee_max, lg_imposee_min, coef_a,
                 ponder_f_roue, nb_val);

$ definition of a type 2 vehicle wheel
<coef_a = 6.00>
ROUE_VEHICULE   (roue_2, roue_cons, impact_x, impact_y, longueur_impo,
                 lg_min, lg_imposee_max, lg_imposee_min, coef_a,
                 ponder_f_roue, nb_val);

$ definition of a type 1 vehicle axle
<ponder_f_essi = 1.00>
<nb_roues       = 2>
<y_roue1        = 0.25>
<y_roue2        = 2.25>
ESSIEU_VEHICULE   (essieu_1, ponder_f_essi, nb_roues)
                  (roue_1, roue_1) (y_roue1, y_roue2);

$ definition of a type 2 vehicle wheel
ESSIEU_VEHICULE   (essieu_2, ponder_f_essi, nb_roues)
                  (roue_2, roue_2) (y_roue1, y_roue2);

$ definition of a type 1 vehicle
<ponder_f_vehi = 0.00>
<dist_avant    = 2.25>
<dist_apres    = 2.25>
<enleve_avant  = 0.00>
<enleve_apres  = 0.00>
<nb_essieux    = 3>
<x_essieu1     = 0.00>
<x_essieu2     = 1.50>
<x_essieu3     = 6.00>
VEHICULE  (vehicule_1, ponder_f_vehi, laisser_unif, dist_avant,
           dist_apres, enleve_avant, enleve_apres, nb_essieux,
          (essieu_2, essieu_2, essieu_1)
          (x_essieu1, x_essieu2, x_essieu3);

$ definition of a type 1 vehicle convoy
<nb_vehicules = 2>
CONVOI_VEHICULES  (convoi_1, convoi_inde_2, pr_inutilise,
                   nb_vehicules) (vehicule_1, vehicule_1);
```

```
$ definition of the type 1 concentrated load
<ponder_d_conc = 1.00>
<ponder_f_conc = 0.00>
<coef_rech     = 0.50>
CONCENTREE  (charge_conc_1, vehicule_cent, convoi_1,
             ponder_d_conc, ponder_f_conc, coef_rech);

$ definition of the load applicable to a type 1 platform
CHARGE_VOIE  (charge_1, charge_conc,
              charge_conc_1, no_inutilise, laisser_unif);

$ definition of the platforms
<pas1     = 0.50>
<pas2     = 3.00>
<nb_v_max = 10>
<lvoi_max = 2.50>
<nb_clas  = 3>
<nb_degr  = (nb_v_max)>
VOIES  (voies_1, type_voies_6, degres_glob)  CIRCULATION VOIES
       (pas1, pas2, nb_v_max, lvoi_max, nb_clas, nb_degr)
       1.20 1.10 0.95 0.80  6*0.70
                           10*1.00
       1.00                 9*0.80;

$ definition of the traffic load conditions
MODALITES  (modalites_1, modalites_gene, largeur_1, longueur_1,
            no_inutilise, voies_1, bande_1);

$ definition of the traffic
<nb_charges = (nb_v_max)>
TRAFIC
'FASCICULE 61 - TITRE II DU C.P.C. - CHARGE Bc - SENS POSITIF'
        (trafic_1, modalites_1, nb_charges) (nb_charges)*(charge_1);

FIN;
```

C.4 - C1 CONVOY OF CIRCULAR 83

```
DEBUT  T_C1P;
$ Circular R/EG.3, 20 July 1983 - C1 convoy - positive direction

$ definition of the loadable length
LONGUEUR_CHARGEABLE  (longueur_1) SUPPORT;

$ definition of the loadable width
LARGEUR_CHARGEABLE  (largeur_1) CIRCULATION CHAUSSEES;

$ definition of the central reservation
BANDE_CENTRALE  (bande_1, bande_n_char) CIRCULATION;

$ definition of the type 1 vehicle wheel
<impact_x       = 0.37>
<impact_y       = 0.37>
<lg_min         = 0.00>
<lg_imposee_max = 1.00>
<lg_imposee_min = 1.00>
<coef_a         = (6.00/2.00)>
<ponder_f_roue  = 1.00>
<nb_val         = 0>
ROUE_VEHICULE  (roue_1, roue_cons, impact_x, impact_y, longueur_impo,
                lg_min, lg_imposee_max, lg_imposee_min, coef_a,
                ponder_f_roue, nb_val);

$ definition of the type 2 vehicle wheel
<impact_x       = 0.37>
<impact_y       = (0.37+0.37)>
<coef_a         = (12.00/2.00)>
ROUE_VEHICULE  (roue_2, roue_cons, impact_x, impact_y, longueur_impo,
                lg_min, lg_imposee_max, lg_imposee_min, coef_a,
                ponder_f_roue, nb_val);

$ definition of the type 3 vehicle wheel
<impact_x       = 0.00>
<impact_y       = 0.00>
<coef_a         = (12.80/4.00)>
ROUE_VEHICULE  (roue_3, roue_cons, impact_x, impact_y, longueur_impo,
                lg_min, lg_imposee_max, lg_imposee_min, coef_a,
                ponder_f_roue, nb_val);

$ definition of the type 1 vehicle axle
<ponder_f_essi = 1.00>
<nb_roues      = 2>
<y_roue1       = 2.00>
<y_roue2       = 4.00>
ESSIEU_VEHICULE  (essieu_1, ponder_f_essi, nb_roues)
                 (roue_1, roue_1) (y_roue1, y_roue2);

$ definition of the type 2 vehicle axle
ESSIEU_VEHICULE  (essieu_2, ponder_f_essi, nb_roues)
                 (roue_2, roue_2) (y_roue1, y_roue2);

$ definition of the type 3 vehicle axle
<nb_roues      = 4>
<y_roue1       = 1.28>
<y_roue2       = 2.10>
<y_roue3       = 3.90>
<y_roue4       = 4.72>
ESSIEU_VEHICULE  (essieu_3, ponder_f_essi, nb_roues)
                 (nb_roues)*(roue_3)
                 (y_roue1, y_roue2, y_roue3, y_roue4);
```

```
$ definition of the type 1 vehicle
<ponder_f_vehi = 0.00>
<dist_avant    = 0.00>
<dist_apres    = 0.00>
<enleve_avant  = 0.00>
<enleve_apres  = 0.00>
<nb_essieux    = 8>
<x_essieu1     = 0.00>
<x_essieu2     = 2.75>
<x_essieu3     = 4.10>
<x_essieu4     = 9.10>
<x_essieu5     = (9.10+1.00*1.55)>
<x_essieu6     = (9.10+2.00*1.55)>
<x_essieu7     = (9.10+3.00*1.55)>
<x_essieu8     = (9.10+4.00*1.55)>
VEHICULE  (vehicule_1, ponder_f_vehi, laisser_unif, dist_avant,
           dist_apres, enleve_avant, enleve_apres, nb_essieux)
           5*(essieu_3) (essieu_2, essieu_2, essieu_1)
          (x_essieu1, x_essieu2, x_essieu3, x_essieu_4,
           x_essieu5, x_essieu6, x_essieu7, x_essieu_8);

$ definition of a type 1 vehicle convoy
<nb_vehicules = 1>
CONVOI_VEHICULES  (convoi_1, convoi_inde_1, pr_inutilise,
                   nb_vehicules, vehicule_1);

$ definition of the type 1 concentrated load
<ponder_d_conc = 1.00>
<ponder_f_conc = 0.00>
<coef_rech     = 0.90>
CONCENTREE  (charge_conc_1, vehicule_inte, convoi_1,
             ponder_d_conc, ponder_f_conc, coef_rech);

$ definition of the load applicable to a type 1 platform
CHARGE_VOIE  (charge_1, charge_conc,
              charge_conc_1, no_inutilise, laisser_unif);

$ definition of the platforms
<nb_clas          = 3>
<nb_chaussees_max = 2>
<nb_degr          = (nb_chaussees_max)>
VOIES  (voies_1, type_voies_5, degres_voie)  CIRCULATION CHAUSSEES
       (nb_clas, nb_degr) 1.00 0.00  1.00 0.00  1.00 0.00;

$ definition conditions of traffic load
MODALITES  (modalites_1, modalites_gene, largeur_1, longueur_1,
            no_inutilise, voies_1, bande_1);

$ definition of the traffic
<nb_charges = (nb_chaussees_max)>
TRAFIC
'CIRCULAIRE R/EG.3 DU 20 JUILLET 1983 - CONVOI C1 - SENS POSITIF'
        (trafic_1, modalites_1, nb_charges) (nb_charges)*(charge_1);

FIN;
```

C.5 - MODEL 1, EUROCODE 1 CLASS 1

```
DEBUT  E_M1C1_C;
$ Eurocode 1 - Road loads - Model 1 - Class 1 - characteristics

$ definition of the loadable length
LONGUEUR_CHARGEABLE  (longueur_1) SUPPORT;

$ definition of the loadable width
LARGEUR_CHARGEABLE  (largeur_1) CIRCULATION CHAUSSEES;

$ definition of the central reservation
BANDE_CENTRALE  (bande_1, bande_n_char) CIRCULATION;

$ definition of a type 1 vehicle wheel
<impact_x        = 0.40>
<impact_y        = 0.40>
<lg_min          = 0.00>
<lg_imposee_max = 1.00>
<lg_imposee_min = 1.00>
<coef_a          = 15.00>
<ponder_f_roue   = 1.00>
<nb_val          = 0>
ROUE_VEHICULE   (roue_1, roue_cons, impact_x, impact_y, longueur_impo,
                 lg_min, lg_imposee_max, lg_imposee_min, coef_a,
                 ponder_f_roue, nb_val);

$ definition of a type 1 vehicle axle
<ponder_f_essi = 1.00>
<nb_roues       = 2>
<y_roue1        = 0.50>
<y_roue2        = 2.50>
ESSIEU_VEHICULE  (essieu_1, ponder_f_essi, nb_roues)
                 (roue_1, roue_1) (y_roue1, y_roue2);

$ definition of a type 1 vehicle
<ponder_f_vehi = 0.00>
<dist_avant     = 0.20>
<dist_apres     = 0.20>
<enleve_avant  = 1.00>
<enleve_apres  = 1.00>
<nb_essieux     = 2>
<x_essieu1      = 0.00>
<x_essieu2      = 1.20>
VEHICULE  (vehicule_1, ponder_f_vehi, laisser_unif, dist_avant,
           dist_apres, enleve_avant, enleve_apres, nb_essieux)
          (essieu_1, essieu_1) (x_essieu1, x_essieu2);

$ definition of a type 1 vehicle convoy
<nb_vehicules = 1>
CONVOI_VEHICULES  (convoi_1, convoi_inde_1, pr_inutilise,
                   nb_vehicules, vehicule_1);

$ definition of a type 1 concentrated load
<ponder_d_conc = 1.00>
<ponder_f_conc = 0.00>
<coef_rech      = 0.90>
CONCENTREE  (charge_conc_1, vehicule_droi, convoi_1,
             ponder_d_conc, ponder_f_conc, coef_rech);

$ definition of a type 2 concentrated load
<ponder_d_conc = (2.00/3.00)>
CONCENTREE  (charge_conc_2, vehicule_droi, convoi_1,
             ponder_d_conc, ponder_f_conc, coef_rech);

$ definition of a type 3 concentrated load
<ponder_d_conc = (1.00/3.00)>
CONCENTREE  (charge_conc_3, vehicule_droi, convoi_1,
             ponder_d_conc, ponder_f_conc, coef_rech);
```

```
$ definition of a type 1 uniform load density
<lg_min         = 0.00>
<coef_b         = 0.90>
<coef_c         = 0.00>
<coef_d         = 0.00>
<lg_imposee_max = 1.00>
<lg_imposee_min = 1.00>
<nb_val         = 0>
DENSITE_UNIFORME  (densite_1, densite_cons, lg_min, pr_inutilise,
                   coef_b, coef_c, coef_d,
                   lg_imposee_max, lg_imposee_min, nb_val);

$ definition of a type 2 uniform load density
<coef_b         = 0.25>
DENSITE_UNIFORME  (densite_2, densite_cons, lg_min, pr_inutilise,
                   coef_b, coef_c, coef_d,
                   lg_imposee_max, lg_imposee_min, nb_val);

$ definition a type 1 uniform load
<ponder_d_unif = 1.00>
<ponder_f_unif = 0.00>
UNIFORME  (charge_unif_1, aires_cumu, densite_1, longueur_calc,
           ponder_d_unif, ponder_f_unif);

$ definition a type 2 uniform load
UNIFORME  (charge_unif_2, aires_cumu, densite_2, longueur_calc,
           ponder_d_unif, ponder_f_unif);

$ definition of the load applicable to a type 1 platform
CHARGE_VOIE  (charge_1, charge_gene,
              charge_conc_1, charge_unif_1, laissser_unif);

$ definition of the load applicable to a type 2 platform
CHARGE_VOIE  (charge_2, charge_gene,
              charge_conc_2, charge_unif_2, laissser_unif);

$ definition of the load applicable to a type 3 platform
CHARGE_VOIE  (charge_3, charge_gene,
              charge_conc_3, charge_unif_2, laissser_unif);

$ definition of the load applicable to a type 4 platform
CHARGE_VOIE  (charge_4, charge_unif,
              no_inutilise, charge_unif_2, laissser_unif);

$ definition of platforms
<pas      = 3.00>
<nb_voies = 3>
<lvoi_max = 3.00>
<nb_clas  = 3>
<nb_degr  = 4>
VOIES  (voies_1, type_voies_4, degres_g_glob)  CIRCULATION LA_VOIES
       (pas, nb_voies, lvoi_max, nb_clas, nb_degr)
       4*1.00  4*1.00  4*1.00;

$ definition of the traffic load conditions
MODALITES  (modalites_1, modalites_gene, largeur_1, longueur_1,
            no_inutilise, voies_1, bande_1);

$ definition of the traffic
<nb_charges = 4>
TRAFIC
'EUROCODE 1 - CHARGE ROUTIERE - MODELE 1 - CLASSE 1 - CARACTERISTIQUE'
        (trafic_1, modalites_1, nb_charges)
        (charge_1, charge_2, charge_3, charge_4);

FIN;
```

Annex D

Methodological advice

This annex will be subsequently developed; refer to the corresponding part of the former Reference Manual.

Annex E

Railway loads

This annex is intended to present the approach aiming to introduce actions on bridges related to rail traffic in the PCP calculator. It is important to note that the load models defined below are from European regulations and do not describe actual loads. "They were selected so that their effects, dynamic increases being considered separately, are those of the actual traffic." Actions due to rail traffic are given for:

- Static vertical loads: load models 71, SW/0, SW2, "empty train"
- Static horizontal loads: centrifugal force, nosing forces, acceleration and braking forces
- Static vertical fatigue loads
- Dynamic actions due to passing trains

Static fields are implemented in the form of PCP traffic component actions that the user can configure. All of these models are compiled into a single file: **trafic.don**. Dynamic fields are defined in separate files that will be called by the user.

Parts E.2 to E.5 are constructed according to the same pattern: *Presentation, Interpretation, Implementation with traffic data and Use.* The first section is intended to serve as a reminder of the regulations concerning the load conditions. The second looks at certain points of the standard that require interpretation or further calculation. The third explains the approach implemented in the definition of the traffic model as well as the name of the model. The *Use* section informs the user about the potential parameters to be changed. The last part presents examples of practical uses for each load model discussed in the previous 4 parts.

E.1 - DOCUMENTS USED

- Eurocode 1 "Actions on structures", Part 2: "Actions on bridges due to traffic"; NF ZN 1991-2
- Booklet 2.01 Standard SNCF Requirements: "Design and calculation rules for concrete, steel or composite structures"
- The bracing of composite multi-beam railway bridges, Yannick Sieffert, 2004, Thesis, INSA Lyon
- Chapter 10 of the PCP instructions: "Traffic definition"
- Chapter 11 of the PCP instructions: "Dynamic Study"

E.2 - PRIOR INFORMATION

All traffic defined below can be found in the example case directory entitled **pont_ferr:**

- trafic.don contains all the static load models
- HSLM-A01.don, HSLM-A02.don, etc. contain the load models for the dynamic study

These files are not intended to be executed by PCP. When calling the traffic model with the overload file ACTION command (Chapter 9 of the PCP instructions), PCP automatically retrieves their definitions from the trafic.don file located in the case directory. Files can be copied to the directory of another case for reuse.

Furthermore, any rail traffic model call requires the definition of a CHAUSSEE, and one or more VOIES in the CIRCULATION command.

N.B. : the case example pont_ferr presents a straight structure. Load conditions involving centrifugal force are not processed. Another case example, entitled **etude_ferrov**, presents the use of three load models involving centrifugal force.

E.3 - STATIC VERTICAL LOADS

E.3.1 – Load model 71

Presentation

Load model 71 represents the static effect of vertical loading due to standard railway traffic. (EC1-2 §6.3.2)

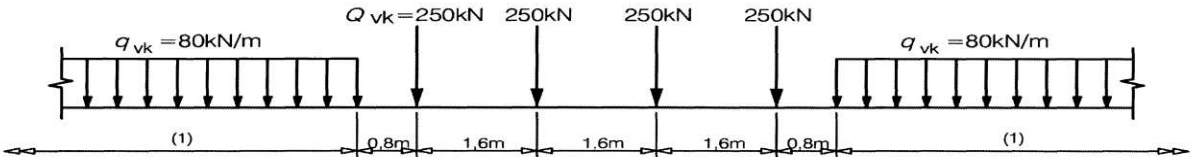


Figure 1 - Load model 71

The characteristic values of Figure 1 must be multiplied by a coefficient α on lines where rail traffic circulation is heavier or lighter than normal. When multiplied by the coefficient α, the loads are called "classified vertical loads". The coefficient α must be chosen from the following values:

0.75 – 0.83 – 0.91 – 1.00 – 1.10 – 1.21 – 1.33 – 1.46 (EC1-2 §6.3.2)

For international freight lines, the α coefficient is taken to be 1.33. On the rest of the rail network at standard spacing, the α coefficient is set equal to 1.00. (EC1-2 National Annex Clause 6.3.2 (3)).

The load pattern 71 may be split (CPC, booklet 2.1 §1.3.1.2.1).

Longitudinal distribution

According to §6.3.6.2 of EC1-2, "point loads of load model 71 [...] can be uniformly distributed in the longitudinal direction".

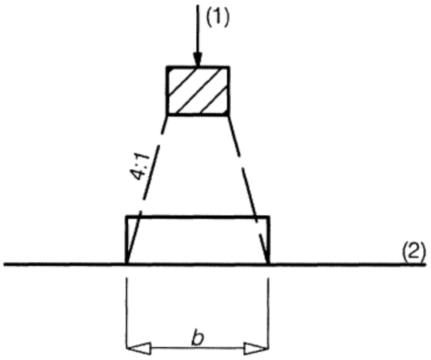
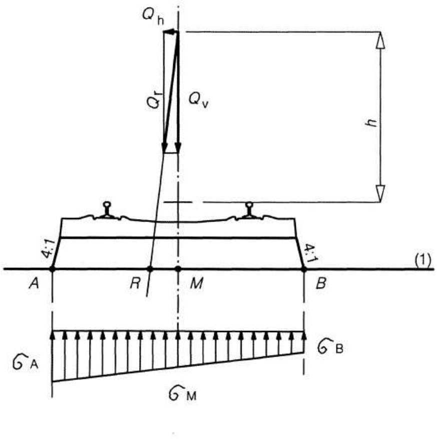


Figure 2 - Longitudinal distribution by sleepers

Transverse distribution

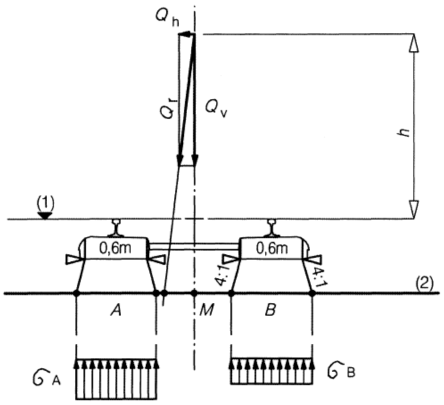
The transverse distribution depends on the type of sleepers used on the track:

- Monoblock sleeper



`Figure 3 - Distribution of loads by sleepers and ballast`

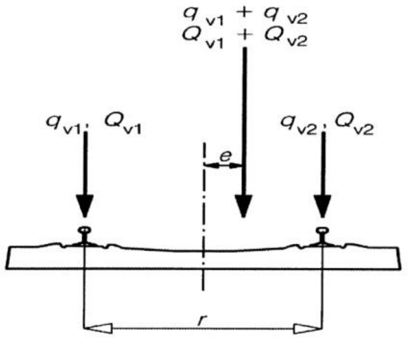
- Bi-block sleeper



`Figure 4 - Distribution of loads by sleepers and ballast`

Eccentricity of vertical loads

The effect of lateral displacement of the loads must be considered (load models 71 and SW/0). The eccentricity of vertical loads can be neglected when considering fatigue.



$$q_{v1}, q_{v2}, Q_{v1}, Q_{v2} = (1)$$

$$q_{v1} + q_{v2}, Q_{v1} + Q_{v2} = (2)$$

$$\frac{q_{v2}}{q_{v1}}, \frac{Q_{v2}}{Q_{v1}} \leq 1,25$$

$$e \leq \frac{r}{18}$$

$$r = (3)$$

Key

(1) Uniform linear load and point loads on each of the rails, as appropriate
(2) LM 71 (and SW/0 if necessary)
(3) Transverse distance between wheel loads

Application

For the determination of the most unfavorable effects resulting from the application of load model 71:

- The uniform linear load q_{vk} must be applied to one track over as many lengths as necessary; up to four individual concentrated loads Q_{vk} must be applied simultaneously on a track
- For structures supporting two tracks, the load model 71 must be applied to one or the other of the two tracks, or to both
- For structures supporting three or more tracks, load model 71 should be applied in turn to each track or each possible group of two tracks or 0.75 LM71 should be applied to three or more tracks

Interpretation

The uniform linear load q_{vk} must be applied to one track over as many lengths as necessary; up to four individual concentrated loads Q_{vk} must be applied simultaneously on a track. The regulation specifies that the load model 71 may be split. Thus, if a configuration with 1, 2 or 3 axles spaced 1.6 m apart is more unfavorable, this will be the one chosen. On the other hand, it is not intended to vary the axle spacing parameter.

Longitudinal distribution

Sleeper dimensions are generally, for the SNCF, 2.6 m long, 25 cm wide and 15 cm thick. On the other hand, the thickness of ballast (through which the distribution occurs) is a function of the load and the intensity of the traffic. The minimum is 25 cm on conventional lines and 45 cm on high-speed lines. In practice, the thicknesses measured on the existing structures are between 10 cm and 1 m. There is therefore a longitudinal impact on the point forces of:

$$0{,}25 + \frac{thickness_{ballast}}{2}$$

Transverse distribution

- Monoblock sleeper: our model includes only "one-wheel" vehicles and the distribution occurs over a 2.6 m wide sleeper:

$$2{,}6 + \frac{thickness_{ballast}}{2}$$

- Bi-block sleeper: in this case, our model includes two-wheel vehicles and the distribution occurs over two 0.6 m wide blocks:

$$0{,}6 + \frac{thickness_{ballast}}{2}$$

Implementation with traffic data

Traffic is named: **T_TM71** (**T** = Traffic; **T**=Train; **M71** = Model 71).

PCP applies from 0 to 4 point loads, spaced by 1.6 m, as shown in the diagram in §6.3.2 EC1-2 . If one of the charges is favorable, it is not positioned, but its footprint is retained. We choose to keep the footprint so that no uniform load is positioned there by PCP.

The uniform load is applied "over as many lengths as necessary" to obtain the most unfavorable configuration. For this, the "aires_cumu" option is selected when defining the uniform load in PCP. Thus, all areas with an unfavorable sign are integrated and combined globally, without considering the combinations of elementary areas. The calculated conventional loaded length is the average length of the calculated areas.

Important Note: The coefficient α is **never** taken into account in the calculations. It is to be taken into account when the combinations of actions are determined by the user.

With this first model, it is not possible to take account of the eccentricity of the vertical loads, since the uniform charge is applied over the entire width of the track. A second model has therefore been defined, where uniform load is approximated by wheels with a length of 1 m. This traffic is named: **T_TM71_2**. This therefore involves defining a track wider than the vehicle, and PCP moves the traffic transversely to find the most unfavorable position.

Use

The user must define two parameters in the traffic definition file:

- epais_ball: is the track ballast thickness. In France, the thickness of ballast depends on the load and traffic intensity. It is advisable to have a ballast thickness of at least 25 cm on conventional lines and 45 cm on high-speed lines.
- wheels: 1 or 2. This parameter characterizes the transverse distribution through the ballast. For monoblock sleepers, select 1. For bi-block sleepers, select 2.

These two parameters are already present in the file. They are to be defined by the user according to the project.

E.3.2 - Load model SW/0

Presentation

Load model SW/0 represents the static effect of vertical loading due to a standard rail traffic on continuous beams.

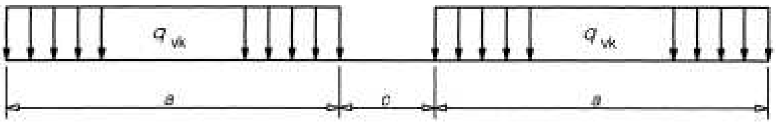


Figure 5 - Load model SW/0

Where:

- $q_{vk} = 133$ kN/m
- a = 15.0 m
- c = 5.3 m

According to sections 1.3.1.3 and 1.3.1.5 of CPC booklet 2.01, load pattern SW/0 should not be split. Eurocode provides no guidance on the subject.

Interpretation

Since the load model SW/0 cannot be split, we are obliged to respect the application length of the two distributed loads, and the distance between them.

Implementation with traffic data

The traffic is named: **T_TSW0**.

Given the constraints, we have chosen to model this load condition by vehicle wheels: each distributed load is composed of 15 wheels with a length of 1 m. The inherent load at each of the wheels is constant: 133 kN

The spacing between the axes of the two wheels of the same axle is:

$$\textit{Rails spacing} + 2 \cdot \left(\frac{\textit{Wheel width}}{2}\right) = 1.435 + 2 \cdot \left(\frac{0.1}{2}\right) = 1.535\text{m}$$

<u>Note:</u> Due to the nature of the non-split load model, it is important to set the favorable weighting coefficients to 1 for the wheels and axles. Only that of the vehicle should be equal to 0.

The No. 5 track type is chosen because in our case the number of tracks is imposed and their positions and widths are known (for the definition of the different track types, refer to paragraph 10.14 of the PCP manual, available to download on the Sétra engineering structures software website).

Use

No user defined parameters.

E.3.3 - Load model SW/2

Presentation

The model SW/2 is very similar to SW/0:

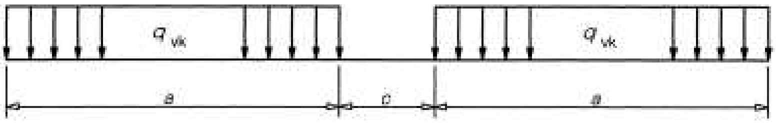


Figure 6 - Load model SW/2

Where:

- q_{vk} = 150 kN/m
- a = 25.0 m
- c = 7.0 m

Interpretation

Same remarks as for SW/0.

Implementation with traffic data

The traffic is named: **T_TSW2**.

The inherent load at each wheel is constant: 150 kN

Use

No user defined parameters.

E.3.4 - "Empty train" load model

Presentation

Some specific checks involve a load model called "empty train" which consists of a uniform linear vertical load, with a characteristic value of 10.0 kN/m.

The uniform load is applied "over as many lengths as necessary" to obtain the most unfavorable configuration. We choose the "aires_cumu" option when defining the uniform load in PCP. In this way, all areas with an unfavorable sign are integrated and combined globally, without considering the combinations of elementary areas. The calculated conventional loaded length is the average length of the calculated areas.

N.B.: The "empty train" load model should only be considered for the calculation of structures supporting a single track.

Interpretation

The loading model is applied as is in traffic.

Implementation with traffic data

The traffic is named: **T_TTAV**.

The model consists of a distributed load at constant density.

Use

No user defined parameters.

E.4 - HORIZONTAL STATIC LOADS

E.4.1 - Centrifugal force

Presentation

When the track is curved over the whole or part of the length of a bridge, the centrifugal force and the track cant must be taken into account.

The centrifugal forces are considered as acting horizontally outward, at a height of 1.80 m above the running surface.

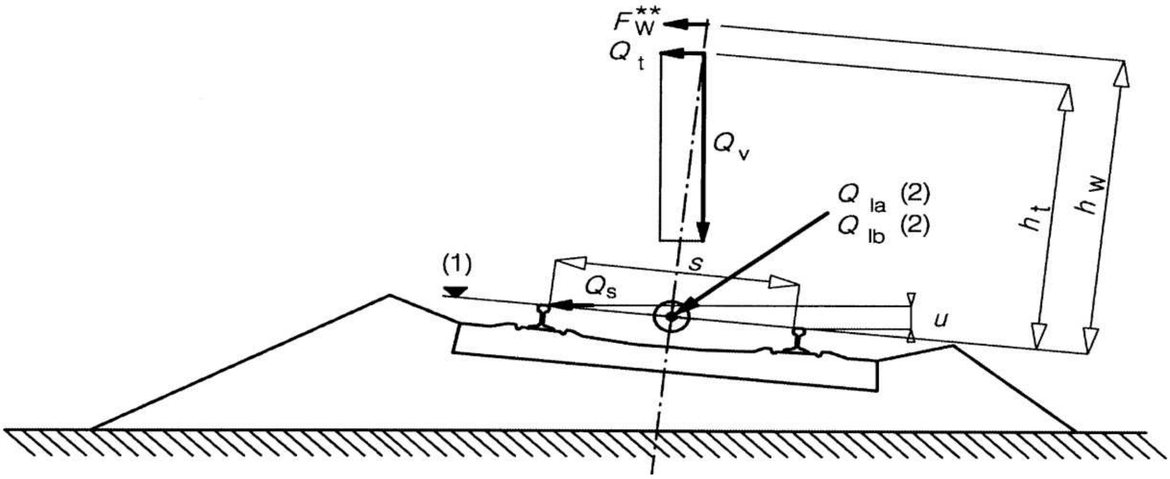


Figure 7 - Symbols and specific dimensions for railway tracks

For certain types of traffic, for example for containers on two levels, a higher value for h_t should be specified. The national annex specifies that the value ht = 2.00 m should be taken over the entire rail network.

The centrifugal force must always be combined with the vertical traffic load. It should not be multiplied by the dynamic coefficient ϕ_2.

The characteristic value of the centrifugal force should be determined using the following formulas:

$$\begin{cases} Q_{tk} = \dfrac{v^2}{g \cdot r} \cdot (f \cdot Q_{vk}) = \dfrac{V^2}{127 \cdot r} \cdot (f \cdot Q_{vk}) \\ q_{tk} = \dfrac{v^2}{g \cdot r} \cdot (f \cdot q_{vk}) = \dfrac{V^2}{127 \cdot r} \cdot (f \cdot q_{vk}) \end{cases}$$

Where:

- Q_{tk}, q_{tk}: characteristic values of the centrifugal forces
- Q_{vk}, q_{vk}: characteristic values of the vertical loads
- f: reduction coefficient
- v: maximum speed [m/s]
- V: maximum speed [km/h]
- G: gravitational acceleration [9.81 m/s^2]
- r: radius of curvature

In the case of a varying radius curve, appropriate mean values may be selected for r.

For load model 71 (and, where applicable, load model SW/0) and maximum line speed at the point considered greater than 120 km/h, the following cases should be considered:

- load model 71 (and, where applicable, load model SW/0) with the associated dynamic coefficient and the centrifugal force for V = 120 km/h, according to the above equations, with f = 1.
- load model 71 (and, where applicable, load model SW/0) with the associated dynamic coefficient and centrifugal force for the maximum speed V specified, in accordance with the above equations, the coefficient f is given by:

$$f = \left[1 - \frac{V-120}{1000} \cdot \left(\frac{814}{V} + 1.75 \right) \cdot \left(1 - \sqrt{\frac{2.88}{L_f}} \right) \right]$$

Where:

- Lf: length of influence of the loaded part of the curved track on the bridge that is the most unfavorable for the dimensioning of the structural element considered [m]
- V: maximum speed [km/h]
- f = 1: for V ≤ 120 km/h or $L_f \leq 2.88$ m
- f < 1: for 120 km/h < V ≤ 300 km/h and $L_f > 2.88$ m
- $f_{(V)} = f_{(300)}$: For V > 300 km/h and $L_f > 2.88$ m

For load models SW/2 and "empty train", a reduction factor value of 1.0 should be used.

Interpretation

The centrifugal force must be associated with a vertical load. A traffic model has been created for each vertical load combined with the centrifugal force.

The resultant of the vertical load and the centrifugal force (of any model) is to be determined. Q denotes the norm of the vertical load, and C denotes the norm of the centrifugal force. Q denotes both a point load and a distributed load.

$$R = \sqrt{\phi_2^2 \cdot Q^2 + C^2} = \sqrt{\phi_2^2 \cdot Q^2 + \left(\frac{V^2}{127 \cdot r} \cdot f \cdot Q \right)^2}$$

Where α denotes the load classification factor, V the maximum speed on the line, r the radius of curvature and f the reduction factor, which is expressed as follows:

$$f = 1 - \frac{V-120}{1000} \cdot \left(\frac{814}{V} + 1.75 \right) \cdot \left(1 - \sqrt{\frac{2.88}{L_f}} \right) = 1 - \delta \cdot \beta \cdot \left(1 - \frac{\gamma}{\sqrt{L_f}} \right) = 1 - \delta \cdot \beta + \frac{\delta \cdot \beta \cdot \gamma}{\sqrt{L_f}}$$

Loading plan definition

The load support must be set at a height of 2.00 m above the running surface. The angle of inclination θ of the reference frame is calculated for a maximum centrifugal force, that is with f = 1, so that the resultant force is located on the Z axis of the local reference frame. In doing so, we assume that the resulting orientation does not change, only its intensity varies.

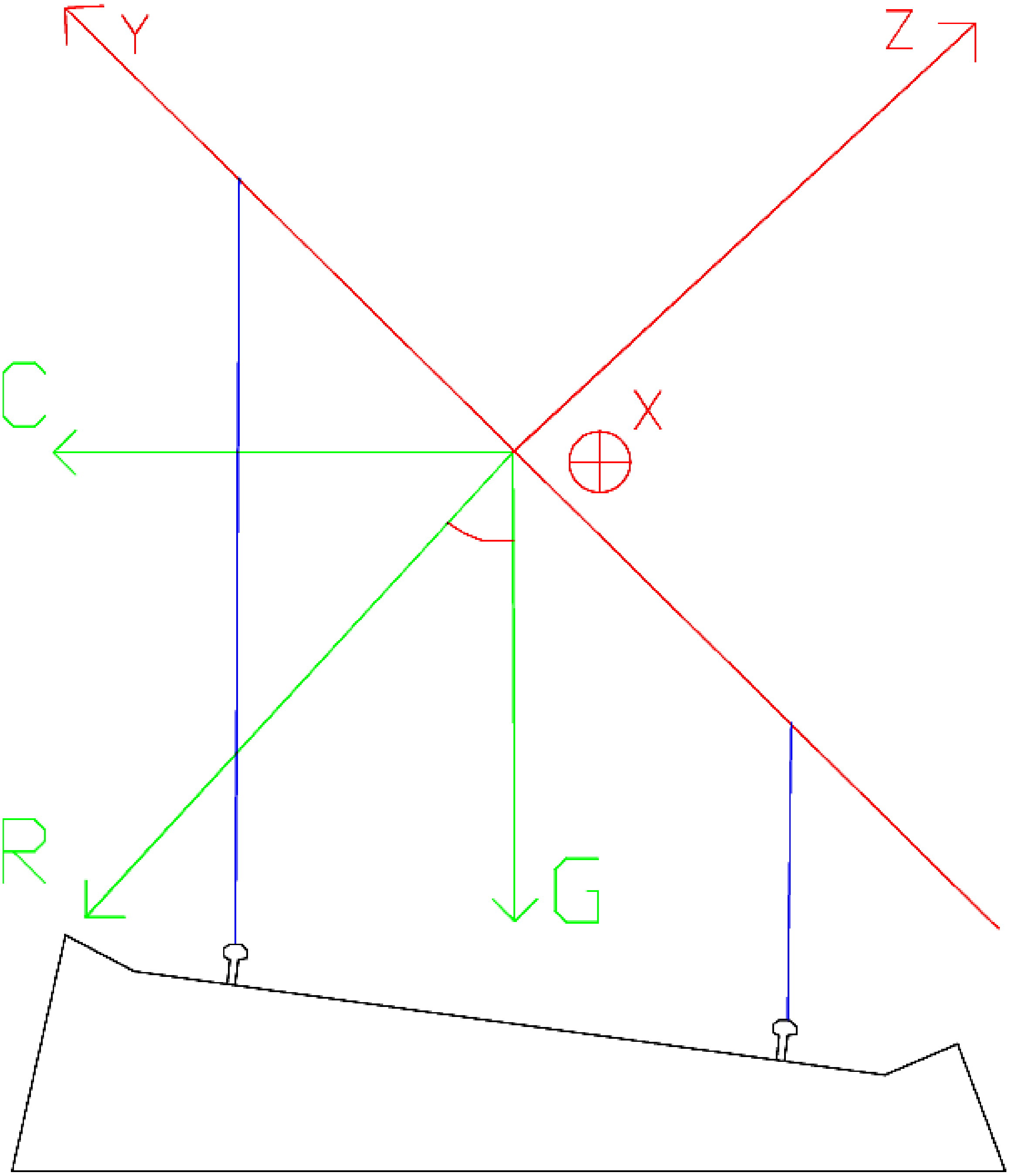


Figure 6 - Definition of the loading plan for the application of the concomitant centrifugal force

Implementation with traffic data

The traffic is named:

LM71 with centrifugal force	**T_TM71C**
SW/0 with centrifugal force	**T_TSW0C**
SW/2 with centrifugal force	**T_TSW2C**

The reduction factor f is 1 if $L_f \leq 2.88$ m or $V \leq 120$ km/h. The use of the MIN function allows us to obtain f = 1 if the user enters a speed lower than 120 km/h.

$$R=Q\cdot\sqrt{\phi_2^2+\left(\frac{V^2}{127\cdot r}\cdot MIN\,(1,1-\delta\cdot\beta+\frac{\delta\cdot\beta\cdot\gamma}{\sqrt{L_f}})\right)}$$

Application to LM71

The operation of the model is the same as in the case of simple static vertical load. The intensity of the load is changed with respect to the vertical static load, and the load support must be consistent with the definition in Figure 6.

We choose the "aires_cumu" option when defining the uniform load in PCP. Thus, all areas with an unfavorable sign are integrated and combined globally, without considering the combinations of elementary areas.

Application to SW/0 and SW/2

As for LM71, we choose to calculate the conventional load length. However, in this case, we have no concomitant uniform charge: calculation of influence lines is impossible. Therefore a fictitious uniform load (whose value is infinitely small) has been added to the model, to make this calculation feasible. Moreover, this uniform load was multiplied by f to obtain the same gradual decrease as for wheel loads. As for LM71, we choose the option "aires_cumu" when defining the uniform load in CFP.

Note: this modeling choice can be a problem in the case of a structure with very short spans. In this case, the user can delete the concomitant uniform load and replace the type_longueur parameter of ROUE_VEHICULE by longueur_impo, and enter its value.

Use

The user must change the following parameters (units are shown in brackets):

- epais_ball: thickness of ballast layer (30 cm in the case of a conventional line, 40 cm for LGV) [m]
- roues: number of wheels of the model. This choice is made based on the sleepers used on the tracks: monoblock: 1 wheel; bi-block: 2 wheels
- coeff_vitesse: maximum speed permitted on the line [km/h]
- longueur_determ: determining length defined in §6.4.5.3 of EC1-2 [m]
- rayon_courbure: line radius of curvature [m]

If the maximum speed of the line at the point of study is greater than 120 km/h, the user must perform an additional study for V = 120 km/h.

Care must be taken to define a loading plan in agreement with the description given in Figure 6. Given this definition, the user must select component No.3 to obtain the gravitational force, and No.4 to get the resultant. To disregard the centrifugal force, simply choose an infinite radius of curvature. However, it is advisable to use the traffic defined in Part 1 if one wishes to disregard the centrifugal force.

E.4.2 - Acceleration and braking forces

Presentation

Acceleration and braking forces act on the upper level of the rails, in the longitudinal direction of the track. They should be considered as linear uniform loads on the effective influence length $L_{a,b}$ for the structural element in question. The direction of the acceleration and braking forces must take account of the direction(s) of travel permitted on each track.

The characteristic values of the acceleration and braking forces shall be taken equal to:

- Acceleration force: $Q_{lak} = 33 * L_{a,b} \leq 1000$ kN for load models 71, SW/0, SW/2 and HSLM
- Braking force: Qlbk = 20 * La,b ≤ 6000 kN for load models 71, SW/0 and HSLM

The characteristic values of the acceleration and braking forces cannot be multiplied by the coefficient ϕ, nor by the coefficient f. However, it is appropriate, for the load models 71 and SW/0, to multiply them by the coefficient a.

Furthermore, acceleration and braking can be neglected for the "empty train" load model.

Interpretation

The choice has been made not to associate the forces of acceleration or braking with a vertical load model in the same traffic definition. These horizontal loads are defined separately.

Implementation with traffic data

They each consist of a distributed load whose intensity varies according to the conventional loaded length. This is calculated by PCP by choosing the "longueur_clac" option when defining the uniform load.

Use

The traffic is named: **T_T_ACC** and **T_T_FRE**.

For acceleration, there is no parameter to be specified by the user.

For braking, the user must complete the value of "coeff_force" according to the vertical load model associated with the braking force: 2.0 for models 71, SW/0 and HSLM and 3.5 for the SW/2 model.

E.4.3 - Nosing forces

Presentation

The nosing force must be considered as a concentrated force acting horizontally on the upper level of the rails and perpendicular to the axis of the track. It must be applied in the case of both a straight track and a curved track. The characteristic value of the nosing force is to be taken as Q_{sk} = 100 kN. It should not be multiplied by the coefficient f. However, it should be multiplied by the coefficient α, if and only if the latter is greater than or equal to 1. The nosing force shall always be combined with a vertical traffic load.

Interpretation

The choice has been made not to associate the nosing forces with a vertical load model in the same traffic definition. This horizontal load is defined separately. In practice, nosing forces are generally used only for local checks.

Implementation with traffic data

From the PCP perspective, this point force is represented by a vehicle with a single wheel. It is possible to reduce the non-zero surface impact of this wheel to a point force, choosing the SIMPLIFIED option when studying the envelopes.

Use

The traffic is named **T_T_LAC.**

No parameters to be modified by the user.

E.5 - FATIGUE

Presentation

All structural elements subjected to stress variations must be evaluated for fatigue damage.

For normal traffic based on the characteristic values of load model 71, including the dynamic factor ϕ, the evaluation of fatigue should be carried out based on traffic combinations of the "standard traffic", "traffic with 250 kN axles"or "light traffic" types, according to whether the structure supports mixed traffic or predominantly heavy or light freight or passenger traffic, according to the specified requirements. Annex D of EC1-2 gives details on the service trains and traffic combinations considered as well as on the dynamic amplification to apply.

When the combination of traffic does not represent the actual traffic (in particular situations, for example, where a limited number of vehicle types dominate the fatigue loads, or by traffic requiring a value of a greater than one), it is necessary to specify a different traffic combination.

Each combination is made from a 25,106 ton annual traffic tonnage traversing the bridge on each track.

For structures supporting more than one track, the fatigue loading must be applied to a maximum of two tracks in the most unfavorable positions.

Fatigue damage should be evaluated based on a 100-year structure lifetime.

It is also possible to carry out fatigue checks from special traffic combinations.

It is appropriate to consider the vertical rail traffic actions including dynamic effects and centrifugal forces for assessing fatigue. Generally, nosing and the longitudinal rail traffic actions can be neglected for fatigue evaluation.

According to Annex D of EC1-2, the ϕ_2 dynamic coefficient (recommended throughout the French railway system), applied to the static load model 71 and models SW/0 and SW/2, represents the extreme loading to be considered when defining the constructive provisions for bridge elements. These coefficients would lead to unnecessarily onerous provisions if applied to real trains used to assess damage resulting from fatigue.

To account for the average effect on the life of the structure assumed equal to 100 years, the dynamic amplification for each real train can be reduced to:

$$1+\frac{1}{2}\cdot\left(\phi'+\frac{1}{2}\phi''\right)$$

expression in which ϕ' and ϕ" are defined below.

The following equations are simplified forms of more general equations, but are sufficiently precise to be used to calculate damage resulting from fatigue. They are valid for maximum permitted vehicle speeds less than or equal to 200 km/h:

$$\phi'=\frac{K}{1-K+K^4}$$

With:

$$K=\frac{v}{160}\ for\ \ L\leq 20m$$

$$K=\frac{v}{47.16\cdot L^{0.408}}\ for\ \ L>20\text{m}$$

And

$$\phi''=0.56\cdot e^{\frac{-L^2}{100}}$$

Where:

- V: maximum permitted speed of the convoy [m//s]
- L: critical length L_ϕ [m] in accordance with 6.4.5.3

As stated previously, the evaluation of fatigue should be carried out based on traffic combinations of the "standard traffic", "traffic with 250kN axles" or "light traffic" types, according to whether the structure supports mixed traffic or predominantly heavy or light freight or passenger traffic.

Interpretation

The traditional view, is that in dimensioning, we use the LM71 convoy only.

Implementation with traffic data

Traffic is created for each train type defined in Annex D of EC1-2.

Use

The traffic is named: **T_TF_T1**, **T_TF_T2**, **T_TF_T3**, etc., **T_TF_T12**.

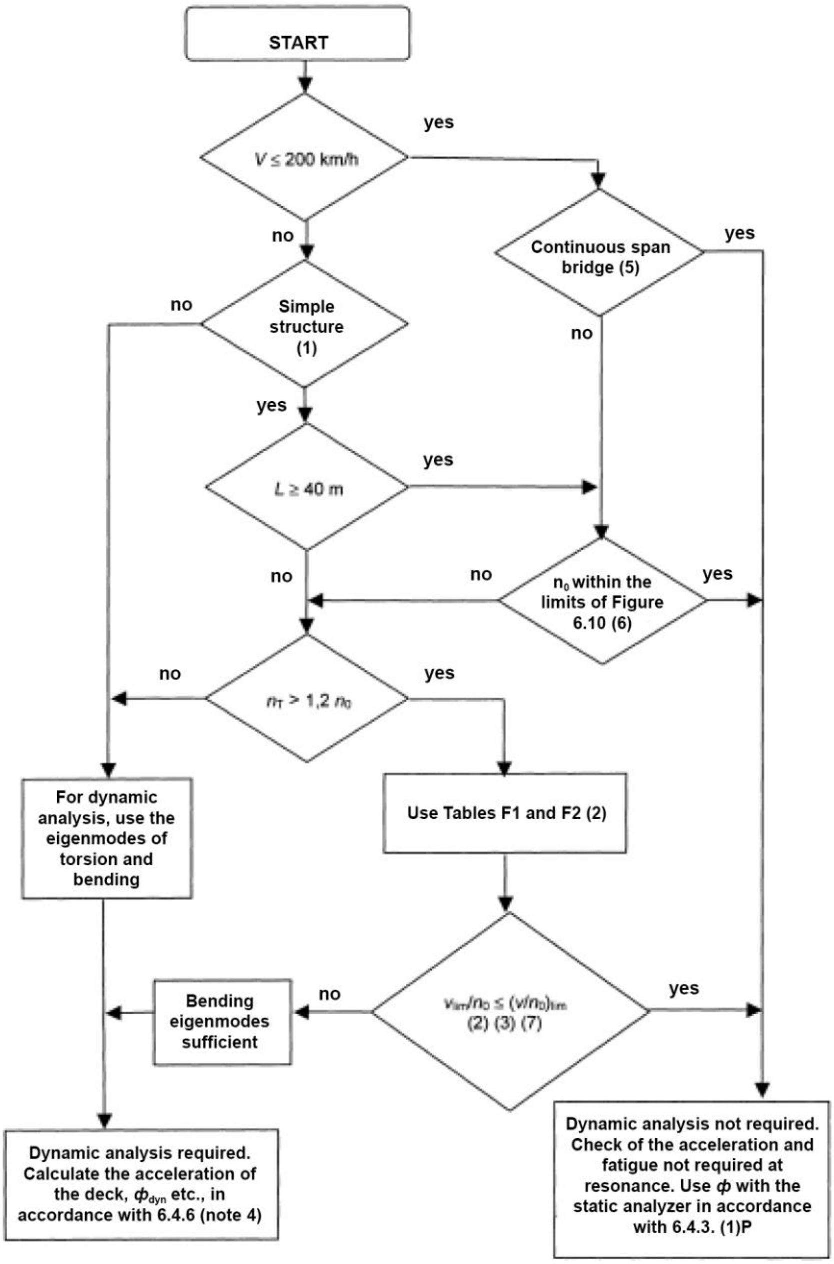
No parameters to be modified by the user.

0.6 - DYNAMIC STUDY

Presentation

A static analysis must be carried out with load models LM71, SW/0 and SW/2 (and possibly the "empty train" load model). The results are multiplied by the dynamic coefficient f2 (which is the dynamic coefficient applicable according to the National Annex), and if required, the coefficient a.

The next step is to determine, using the chart provided by EC1-2, if a dynamic analysis is necessary:



If it turns out that a dynamic analysis is required:

1. The additional load conditions for the dynamic analysis shall conform to §6.4.6.1.2 of EC1-2.

§6.4.6.1.2 of EC1-2 states that "the dynamic analysis shall be performed using the loads indicated in 6.4.6.1.1 (1) and (2) and, where applicable, 6.4.6.1 .1 (7)."

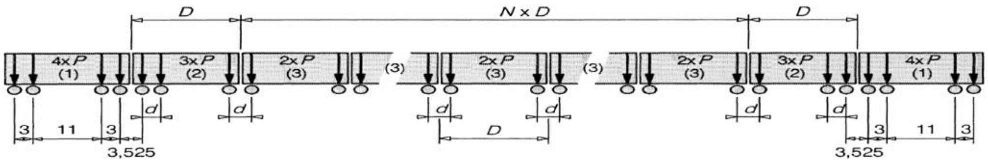
According 6.4.6.1.1 (1): "dynamic analysis shall be performed using the characteristic values of real trains' specified loads. The choice of real trains must consider each of the authorized or contemplated train compositions for each high-speed train type accepted by, or intended for, circulation on the structure at speeds greater than 200 km/h."

Paragraph 6.4.6.1.1 (2) specifies that dynamic analysis "should also be done using the HSLM load model on bridges designed for international lines when European high speed interoperability criteria apply."

In these two paragraphs, the National Annex states that in "the absence of real trains to be considered for dynamic analysis in the individual project, it is advisable to perform this analysis using the HSLM load model." Moreover, it is "recommended to use the HSLM load model, even in the absence of interoperability criteria for the line concerned."

HSLM-A model

The HSLM-A load model is defined as follows:



(1) Locomotive (identical head and tail locomotives)
(2) End wagon (identical head and tail wagons)
(3) Intermediate wagon

Representative train	Number of intermediate wagons *N*	Wagon length *D* [m]	Distance between axes of bogie axles *d* [m]	Point force *P* [kN]
A1	18	18	2,0	170
A2	17	19	3,5	200
A3	16	20	2,0	180
A4	15	21	3,0	190
A5	14	22	2,0	170
A6	13	23	2,0	180
A7	13	24	2,0	190
A8	12	25	2,5	190
A9	11	26	2,0	210
A10	11	27	2,0	210

Figure 7 – HSLM-A model

HSLM-B model

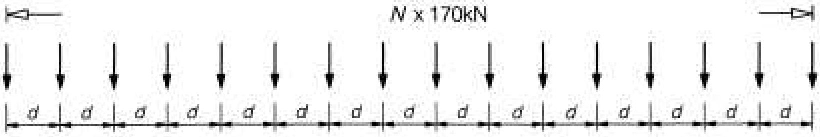
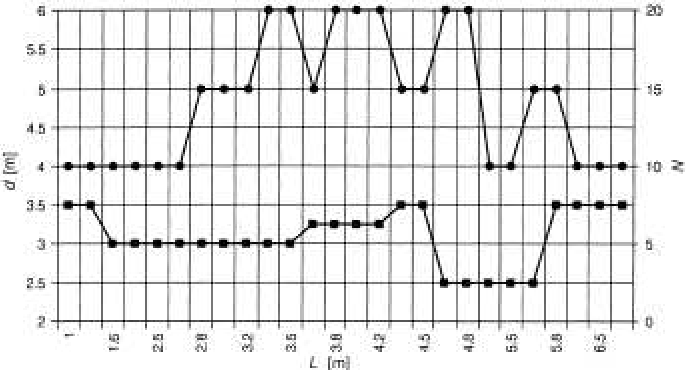


Figure 6.13 — HSLM-B



The HSLM-A model, or the HSLM-B model, should be applied based on the criteria of the following table:

Configuration of the structure	Length	
	$L < 7$ m	$L \geq 7$ m
Span with single supports [a)]	HSLM-B [b)]	HSLM-A [c)]
Continuous structure [a)] or Complex structure [e)]	HSLM-A Trains A1 to A10 included [d)]	HSLM-A Trains A1 to A10 included [d)]

a) Applicable for bridges whose behavior is similar to that of a straight beam (longitudinal direction) or a simple plate on fixed supports, with negligible bias effects.

b) For spans with single supports and a length of less than 7 m, the analysis can be performed using an HSLM-B model unique critical representative train, as indicated in 6.4.1.1(5).

c) For spans with single supports and a length greater than or equal to 7 m, the analysis can be performed using an HSLM-A model unique critical representative train, as indicated in Annex E (or you can use representative trains A1 to A10 included).

d) Use all the trains A1 to A10 for the calculation.

e) Any structure that does not meet the criteria of Note a above. For example: bias structure, structure with significant torsion behavior, structure with side doors with deck vibration modes and significant main beams, etc. In addition, HSLM-B should also be used for complex structures with significant deck vibration modes (structures with thin deck side beams, for example).

NOTE The National Annex or the individual project may specify additional requirements regarding the application of HSLM-A and HSLM-B models to continuous and complex structures.

In the case of passenger trains, taking into account the dynamic effects is valid for the maximum permitted speed of the vehicle up to 350 km/h.

Interpretation

When dynamic analysis is required, the user must apply the HSLM load model systematically, in addition to actual train loads specified by the project.

However, dynamic analyses are usually not made with more than one loaded track, given the low probability of occurrence and unfavorable phasing of signals.

In terms of actual trains, it is customary to replace the axle point loads by 3 loads spaced 60 cm apart (distance corresponding to the interval between two sleepers), with respectively 25%, 50% and 25% of the load.

Implementation with traffic data

A load model was defined for each of the representative trains. The ACTION DETERMINISTE command was used.

Pseudo-programming was used, to describe the load models needed to achieve a dynamic analysis.

Use

To call one of these load models, simply use the LIRE command. The data file must be in the case directory. They are named: **HSLM-A1.don, HSLM-A2.don, HSLM-A3.don, etc., HSLM-A10.don** and **HSLM-B.don**.

The user must set the following parameters in the calling file, and position them before calling the data file containing the representative train:

- speed: maximum permissible speed at the point of study
- sp: 1.20*speed
- direction: direction of circulation of the railway train. Takes the value 1 if the train is traveling in the direction of increasing abscissa, and -1 if it is traveling in the direction of decreasing abscissa
- no_supp: load support number
- pos_trans: transverse positioning of the load model
- length: for the HSLM-A dynamic models, this is the full length of the structure. For the HSLM-B model, it is the length of a span.

If several train models are called in the same dynamic load definition file, these parameters should be defined just before the call of the model, for each model (see example in section E.6).

E.7 - EXAMPLES OF USE

The **pont_ferr** case presents the use of static and dynamic railway load models, with the exception of those involving centrifugal force.

The **etude_ferrov** case deals with the load models involving centrifugal force.

Annex F

Bibliography

The following documents can be found in the "notice" repertory of the PCP repertory. All these documents are in French.

Conseils_methodologiques.pdf

Extract of the former notice which presents examples of studies.

Dynamique_spectrale.pdf

Methodological presentation of an earthquake and wind spectral dynamic calculation.

Hypotheses_methodes.pdf

Extract of the former notice which presents assumptions and calculations methods.

Plaquette.pdf

Leaflet of the PCP software. It can be found in English in the PCP directory.

Resal_pcp.pdf

Document which presents automatic calculation of shear force correction by the RESAL effect in PCP.

These_pcp_florent_robert.pdf

Robert, F. (1999). Contribution à l'analyse non linéaire géométrique et matérielle des ossatures spatiales en Génie civil : Application aux ouvrages d'art. Phd : civil engineering. 250p. It discusses geometrical and material non-linear analyses of structure in civil engineering.

Vent_turbulent_théorie.pdf

Scientific notice which presents the spectral theory of turbulent wind calculation.