

**UNIVERSITY OF LIEGE**  
**Department *ArGEnCO***  
***Service Structural Engineering***

**USER'S MANUAL FOR SAFIR 2019**  
**A COMPUTER PROGRAM FOR ANALYSIS OF STRUCTURES**  
**SUBJECTED TO FIRE**

by

Jean-Marc Franssen [1] & Thomas Gernay [2]

January 2019

[1] [jm.franssen@uliege.be](mailto:jm.franssen@uliege.be) – Liege University, Liege, Belgium

[2] [tgernay@jhu.edu](mailto:tgernay@jhu.edu) – Johns Hopkins University, Baltimore, MD, U.S.A.

## TABLE OF CONTENTS

<b>1</b>	<b>INTRODUCTION</b>	<b>3</b>
1.1	General	3
1.2	Analysis Procedure.	4
	1.2.1 Thermal analysis.	4
	1.2.2 Analysis of torsional stiffness of BEAM elements.	7
	1.2.3 Structural analysis at elevated temperature.	7
1.3	Capabilities of SAFIR.	8
	1.3.1 Capabilities concerning the temperature analysis.	8
	1.3.2 Capabilities concerning the torsional analysis.	8
	1.3.3 Capabilities concerning the structural analysis.	8
1.4	Common Features in all Analyses.	10
1.5	SAFIR functions and user defined functions.	11
	1.5.1 General principle.	11
	1.5.2 SAFIR defined functions.	11
	1.5.3 User defined functions.	12
1.6	Sign Conventions.	13
	1.6.1 Global and local axes.	13
	1.6.2 Stresses.	13
1.7	Units.	14
<b>2</b>	<b>INPUT DESCRIPTION.</b>	<b>15</b>
2.1	Input for SAFIR.	15
2.2	General Data for Structural Analysis.	16
2.3	Material Properties.	19
2.4	Convergence Criteria.	20
<b>3</b>	<b>DETAILED INPUT DATA AND FORMAT.</b>	<b>21</b>
3.1	Description and Format of the .IN file for Thermal Analysis.	21
3.2	Description and Format of the eventual file describing the local HASEMI fire(s) for a thermal analysis.	22
3.3	Description and Format of the .IN file for Structural Analysis.	23
3.4	Description and Format of the .IN file for Torsional Analysis.	59
3.5	Structure of the .TEM files used with the BEAM F. E.	66
3.6	Structure of the .TSH files used with the shell F. E.	68
3.7	Structure of the temperature files used with the truss F. E.	72

# 1 INTRODUCTION

## 1.1 General

SAFIR is a special purpose computer program for the analysis of structures under ambient and elevated temperature conditions. The program, which is based on the Finite Element Method (FEM), can be used to study the behaviour of one, two and three-dimensional structures. The program (SAFIR) was developed at the University of Liège, Belgium, and is today viewed as the second generation of structural fire codes developed in Liège, the first generation being another computer program called Computer Engineering of the Fire design of Composite and Steel Structures (CEFICOSS)<sup>1,2</sup>.

As a finite element program, SAFIR accommodates various elements for different idealization, calculation procedures and various material models for incorporating stress-strain behaviour. The elements include the 2-D SOLID elements, 3-D SOLID elements, BEAM elements, SHELL elements and TRUSS elements. The stress-strain material laws are generally linear-elliptic for steel and non-linear for concrete.

The analysis procedure and the program capability are presented in this Chapter. Details of the data files, material properties and cross sectional shapes are presented in Chapter 2. The detail input and format used in the program are given in Chapter 3, while Chapter 4 presents the theory and formulations of the elements available in the program.

## 1.2 Analysis Procedure

Using the program, the analysis of a structure exposed to fire may consist of several steps. The first step involves predicting the temperature distribution inside the structural members, referred to as '*thermal analysis*'. The torsional analysis may be necessary for 3-D BEAM elements, a section subject to warping and where the warping function table and torsional stiffness of the cross section are not available. The last part of the analysis, termed the '*structural analysis*', is carried out for the main purpose of determining the response of the structure due to static and thermal loading. The various stages of analysis are briefly outlined in the following sections.

### 1.2.1 Thermal analysis

This analysis is usually performed while the structure is exposed to fire. For a complex structure, the sub-structuring technique is used, where the total structure is divided into several substructures and a temperature calculation is performed successively for each of the substructures. This kind of situation does arise in a structure where the members are made of different section types, or made of sections submitted to different fire exposures. The thermal analysis is made using 2-D SOLID elements, to be used later on cross sections of BEAM elements or on the thickness of SHELL elements.

#### **a) Temperatures in beams**

The temperature is non-uniform in the sections of the beam, but there is no heat transfer along the axis of the beams. As an example, a frame structure with reinforced concrete columns, pre-stressed main beams and structural steel secondary beams, will require separate temperature analyses for each of these section types. From these analyses, the temperatures across the cross section are obtained and are stored for subsequent structural analysis where these sections are present.

#### **b) Temperatures in shells**

The temperature is non uniform on the thickness of the shell, but there is no heat transfer in the plane of the shell. The temperature analysis is performed on a section having the thickness of the shell and an arbitrary width, 1 cm for example. The node numbering is from 1 to NNODE / 2 from the bottom to the top of the section and, again, from NNODE / 2 + 1 to NNODE for the second row of nodes.

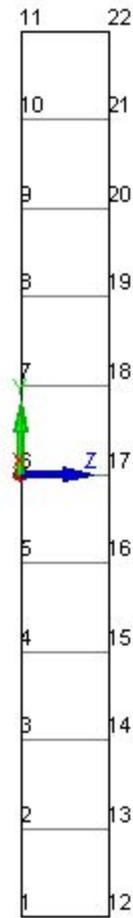


Figure 1 : Model for the thermal analysis of a slab

For example, the Figure above has been created with the following lines for a 10 cm thick slab.

```

NODES
NODE      1  -0.05  0.00
GNODE    11   0.05  0.00   1
REPEAT   11   0.00  0.01   1

```

A .TSH file is created in which the temperatures of the first NNODE / 2 nodes are written. Bellow is given an example of such a file. Note that the temperatures are calculated from  $-t/2$  to  $+t/2$ . For particular cases, like for example a uniform temperature distribution, a similar file can be created with a text editor. The number of elements and the size (here the thickness) of these elements is independent from the number and position of the points of integration that will be used later in the structural analysis. For the structural analysis, the temperatures at the points of integration are linearly interpolated from the temperatures of the nodes.

THIS IS A COMMENT LINE

THICKNESS 0.10

MATERIAL1

REBARS 0

HOT

POSITIONS OF THE NODES.

=====

NUMBER OF POSITIONS: 11

-0.5000E-01 -0.4000E-01 -0.3000E-01 -0.2000E-01 -0.1000E-01 0.0000E+00 0.1000E-01  
0.2000E-01  
0.3000E-01 0.4000E-01 0.5000E-01

TIME= 60.0000 SECONDS OR 1 MIN. 0 SEC.

=====

-0.0500 56.41  
-0.0400 25.16  
-0.0300 20.15  
-0.0200 19.96  
-0.0100 20.00  
0.0000 20.00  
0.0100 20.00  
0.0200 20.00  
0.0300 20.00  
0.0400 20.00  
0.0500 20.00

TIME= 120.0000 SECONDS OR 2 MIN. 0 SEC.

=====

-0.0500 95.76  
-0.0400 41.51  
-0.0300 24.43  
-0.0200 20.54  
-0.0100 20.01

```
0.0000 20.00
0.0100 20.00
0.0200 20.00
0.0300 20.00
0.0400 20.00
0.0500 20.00
```

## 1.2.2 Analysis of torsional stiffness of BEAM elements

This analysis is usually performed when analyzing structures with 3-D BEAM elements, either because non-uniform torsion and beam cross-section were subject to warping (warping function is not equal to zero) or because the torsional stiffness is not available from tables or formulas. The 2-D SOLID elements are used to calculate the warping function and the torsional stiffness of the cross section. The torsional properties obtained from this calculation are added to the results obtained from the temperature analysis of the same cross section for subsequent structural analysis. In cases where the warping function is not necessary, such as in the case of uniform torsion or a cross section with a warping function equal to zero, and if the torsional stiffness can be found in standard tables or by analytical formula, then this analysis need not be performed. In such situations, the torsional stiffness is simply introduced as a property of the cross-section for the structural analysis.

## 1.2.3 Structural analysis at elevated temperature

For each calculation, the loads are applied to the structure, described as BEAM, TRUSS and SHELL elements. The temperature history of the structure, due to fire, is read from the files created during the temperature analysis. As the computation strategy is based on a step-by-step procedure, the following information can be obtained until failure occurs in the structure:

- Displacement at each node of the structure.
- Axial and shear forces and bending moments at integration points in each finite element.
  - Strains, stresses and tangent modulus in each mesh at integration points of each finite element.

## 1.3 Capabilities of SAFIR

SAFIR can be used for performing three different types of calculations, namely, thermal, torsional and structural analysis. The capabilities of the program concerning these three analysis types are outlined in this section.

### 1.3.1 Capabilities concerning the temperature analysis

- Plane sections as well as three-dimensional structures can be analyzed.
- Plane sections are discretized by triangular and/or quadrilateral (rectangular and non-rectangular) elements, allowing representation of virtually all cross sectional shapes.
- Three-dimensional structures are discretized by solid elements (prismatic and non-prismatic) with 6 or 8 nodes. This allows the representation of virtually all structure shapes.
- Variation of material from element to element is possible.
- The fire temperature, defined as a function of time, can either be the standard curves predefined in the code (ISO 834, ASTM E119, ULC S-101) or any other curve can be introduced through data points.
- Cooling down phases can be considered.
- Variation of material properties with temperatures, as well as the evaporation of moisture, can be considered.
- Can analyze thermal performance of materials such as steel, reinforced concrete and composite steel-concrete sections. Other materials can also be analyzed provided their physical properties at elevated temperatures are known.

### 1.3.2 Capabilities concerning the torsional analysis

- Allows virtually all cross section shapes to be represented.
- Materials are considered to be in the elastic stage, at ambient temperature. The user may adjust the obtained torsional stiffness in order to take into

account an increase of temperature during the fire. The torsional stiffness remains constant during the simulation of the structural behaviour.

### 1.3.3 Capabilities concerning the structural analysis

- Plane or 3-D structures can be analyzed.
- The structure is discretized by means of four different element types: Truss elements, made of one single material with one uniform temperature per element; beam elements, shell elements; and solid elements.
- Large displacements are considered in the truss, beam and shell elements.
- The effects of thermal strains (thermal restraint) can be accounted for.
- Material properties are non-linearly temperature dependent.
- Unloading of material is parallel to the elastic-loading branch.
- Local failure of a structural member that does not endanger the safety of the whole structure can be handled by means of a dynamic analysis.
- Nodal coordinates can be introduced in the Cartesian or cylindrical system of axes.
- Imposed displacement (prescribed degrees of freedom) can be introduced.
- Structures with external support inclined at an angle to the global axes can be analyzed.
- Residual stresses (initial strains) can be accounted for.
- Pre-stressed structures can be analyzed.

Automatic adaptation of time step is possible and structural calculation continues until failure or, alternatively, when the maximum deflection has reached a value defined by the user.

## 1.4 Common Features in all Analyses

The common features in all computations are listed as follows:

- Optimization of the matrix in order to reduce the computer storage and calculation time can be performed by the program using internal re-numbering of the system equations. This re-numbering is transparent to the user.
- The same temperature or the same displacement can be imposed at two different nodes by the use of master-slave relations.
- Thermal and mechanical properties of the steel and concrete according to Eurocodes 2, 3 and 4 are embedded in the code and can be used directly.
- Graphic pre-processing and post-processing capabilities are by the SAFIRwizard and DIAMONDXL codes, respectively. When needed, SAFIR could be adapted so as to give the results in a format compatible with commercial graphic software, such as I-DEAS.

## 1.5 SAFIR functions and user defined functions

### 1.5.1 General principle

In different locations of the .IN file, some functions of time can be introduced. They are used either to describe the evolution of the gas temperature in case of a thermal analysis, or to prescribe the evolution of the solution in different nodes and degrees of freedom (either be it a displacement, a temperature or a value of the warping function).

There are two different types of functions:

1. SAFIR defined functions. These functions are embedded in the code. Each function is represented by a name. The comprehensive list is given here bellow.
2. User defined functions. If the name (maximum 10 characters) is not one of the SAFIR defined function, SAFIR will assume that it represents the *filename.filetype* of a file in which the user has described the evolution of the function with time by a series of  $(time_i, value_i)$  pairs (free format). Linear interpolation is made between the defined points. This file describing the function must be in the same folder as the input file.

## 1.5.2 SAFIR defined functions

The comprehensive list of SAFIR defined function is (with  $t$  in seconds):

- F0  $f = 0$
- F1  $f = 1$
- F1PS  $f = t$
- MOINSF1PS  $f = -t$
- FMOINS1PS  $f = -t$
- F2PM  $f = 2t / 60$
- F20  $f = 20$
- F100  $f = 100$
- F1000  $f = 1000$
- F1000PS  $f = 1000t$
- F1PSM1000  $f = 0$  for  $t \leq 1000$   
 $f = t - 1000$  for  $t > 1000$
- F1000THPS  $f = t / 1000$
- FLOAD  $f = t / 20$  for  $t \leq 20$   
 $f = 1$  for  $t > 20$
- FLOAD2  $f = 0.1t - 0.0025t^2$  for  $t \leq 20$   
 $f = 1$  for  $t > 20$
- FISO  $f = 20 + 345 \log_{10}(8t / 60 + 1)$
- FISO0  $f = 345 \log_{10}(8t / 60 + 1)$
- HYDROCARB  $f = 20 + 1080 (1 - 0.325 e^{-0.167t/60} - 0.675 e^{-2.5t/60})$
- ASTM119  $f$  defined by linear interpolation between a set of ( time ; temperature ) pairs, with time in minutes.

Time	Temp.								
0	20								
5	538	65	937	130	1017	250	1100	370	1184
10	704	70	946	140	1024	260	1107	380	1191

15	760	75	955	150	1031	270	1114	390	1198
20	795	80	963	160	1038	280	1121	400	1204
25	821	85	971	170	1045	290	1128	410	1211
30	843	90	978	180	1052	300	1135	420	1218
35	862	95	985	190	1059	310	1142	430	1225
40	878	100	991	200	1066	320	1149	440	1232
45	892	105	996	210	1072	330	1156	450	1239
50	905	110	1001	220	1079	340	1163	460	1246
55	916	115	1006	230	1086	350	1170	470	1253
60	927	120	1010	240	1093	360	1177	480	1260

### 1.5.3 User defined functions

An example of user defined function could be

```
· myfire.fct
```

and the content of the file *myfire.fct* would be:

```
0. 20.
600. 200.
720. 800.
2400. 900.
3600. 300.
7200. 20.
10800. 20.
```

for a natural fire reaching a maximum temperature of 900°C after 40 minutes and decreasing thereafter.

## 1.6 Sign Conventions

The following sign conventions are applied.

### 1.6.1 Global and local axes

Global axes are employed when defining a structure that is to be analyzed using SAFIR. This is done using the Cartesian system of coordinates. For the 2-D (plane) problems, the axes are named G1 and G2, while the local axes are named L1 and L2. Applied force and the displacements are positive in the direction of G1 and G2; the applied moments and rotations are positive in a counter-clockwise direction. For the 3-D problem, the global axes are named G1, G2 and G3 and the local axes are named L1, L2 and L3. The movement G1-G2-G3 is dextrorsum; the applied force and moments, displacements and rotations are all positive in the G1, G2 and G3 directions.

### 1.6.2 Stresses

The stresses are positive in tension. Axial forces, obtained as a summation of the stresses, are also positive in tension. Bending moments in the beam elements, obtained as a summation of  $y_i s_i$ , with  $y_i$  measured on the local axis L1, are positive when fibres having a positive local coordinate are in tension.

## 1.7 Units

The international system of units is adopted. Hence, for instance, all length quantities are in meter. Forces are in Newton. Time is in seconds.

Temperature is in degree Celsius ( $^{\circ}\text{C}$ ).



## 2 INPUT DESCRIPTION

### 2.1 Input for SAFIR

For any analysis using SAFIR, data files acting as input files to the program are prepared. For each analysis type (thermal, torsional or structural analysis), the user prepares one data file. This is an ASCII file, created with a text editor, in a word processor, or by the pre-processor GID (to be obtained separately), and it must have the filetype **.IN**.

This file with a **.IN** extension contains information such as calculation strategy, time discretization, loads, node coordinates, types of finite elements used, material properties, etc. For structural analysis, the **.IN** file specifies the name of the files created during thermal and torsional analyses and in which the temperature data is stored.

Figure 2 shows a schematic representation of the different steps and files that may be involved in the case of a frame structure comprised of two types of different sections, one for the columns and one for the beam. The structure is modelled by beam finite elements. The user must create the **.IN** files. The commands, format and number of lines required for a section in the input files are briefly given in the following sections, whereas the detailed structure of these files is given in Chapter 3.

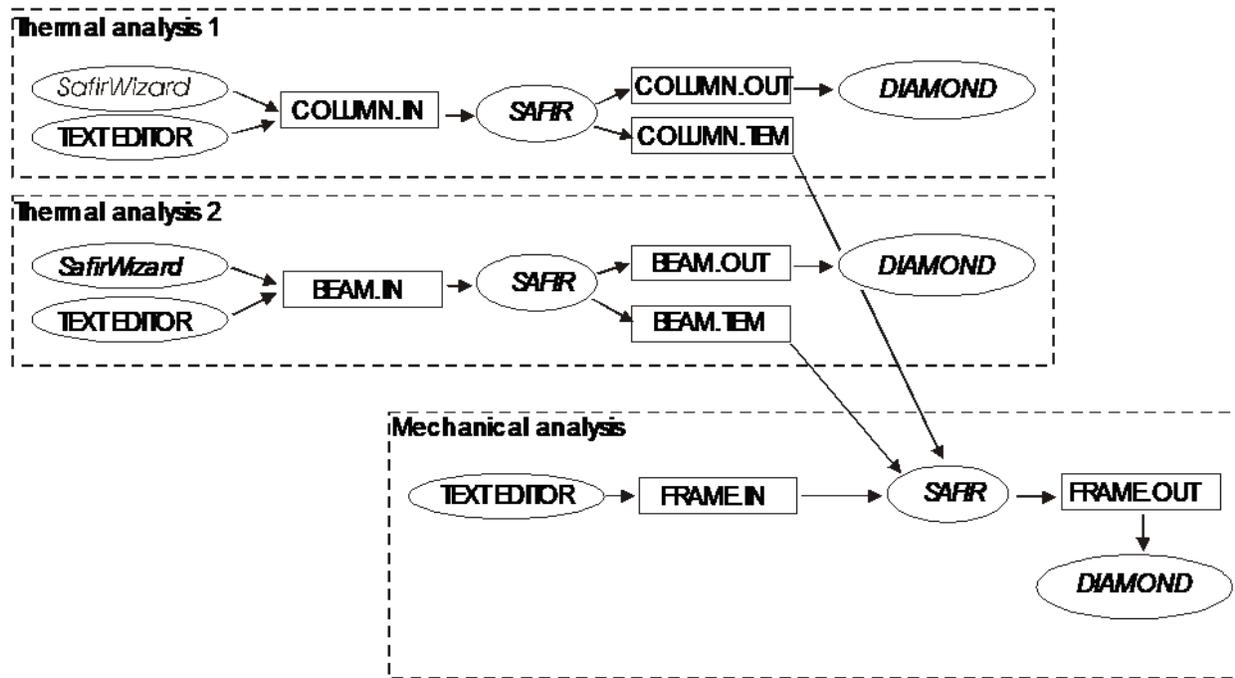


Figure 2 : files and steps

## 2.2 General Data for Structural Analysis

The general data for the **.IN** file of a structural analysis is briefly presented in Table 2. In each input line, a command is given followed by the parameters for the command. Full details of all the commands are given in Chapter 3.

**Table 2: Input data file (.IN) format for structural analysis**

Command	Parameter Format	Notes
	<A80>	Comments, multiple lines possible
	Blank line for end of comments	
"NNODE"		

"NDIM"		
"NDDLMAX"		
"STATIC" OR "DYNAMIC"		The user has to indicate (following the word "STATIC" or "DYNAMIC") the resolution technic choosen. Either "PURE_NR" or "APPR_NR".
"NLOAD"		
"OBLIQUE"		
"COMEBACK" OR "NOCOMEBACK"		Only COMEBACK needs a parameter
"NOENUM" OR "RENUMPERM" OR "RENUMGEO" OR "RENUM" OR "READRENUM"		Only RENUMGEO needs a parameter, either 0 or a node number
"NMAT"		
"ELEMENTS"		
"BEAM"		Optional
"NG"		For beam elements
"NFIBER"		For beam elements
"TRUSS"		Optional
"SHELL"		Optional
"NGTHICK"		For shell elements
"NGREBARS"		For shell elements
"SOLID"		Optional
"NG"		For solid elements

"ENDELEM3	end of ELEMENT section	
"NODES" or "NODES_CYL"		Choose Cartesian or Cylindrical coordinates
"NODE" or "GNODE" or "REPEAT"	REPEAT N1 TO N2 STEP N3 TIMES N4	Multiple lines possible. Last 15 not present if NODE command is used
"FIXATIONS"		
"BLOCK"		Optional, multiple lines possible
"SAME", "SAMEALL" <b>OR</b> "REPEAT"		Optional, multiple lines possible
"END_FIX"	line for end of section	
"NODOFBEAM"		For beam elements
<i>Filename.TEM</i>	A20	Left justified file name of <b>.TEM</b> file for beams
"TRANSLATE"		Multiple lines possible for beams
"END_TRAN"	end of section	
"ELEM" or "GELEM"		Entry to list nodes of all elements and material type
"NODOFSOLID"		Optional
<i>Filename.OUT</i>		File name for <b>.OUT</b> file related to solid elements
"ELEM" or "GELEM" <b>OR</b> "REPEAT"		Multiple lines possible for solid elements
" ENDSYM"		For solid elements
"NODOFSHELL"		Optional
<i>Filename.TSH</i>	<A20>	File name for <b>.TSH</b> file related to shell elements
" TRANSLATE"	<I5><I5>	As many line as necessary for shell elements
"ENDTRANSLA"		End of translation for shell elements

“ELEM” OR “REPEAT”	<9*I5>	Multiple lines possible for shell elements
“NODOFTRUSS”		Optional
<i>Filename.TRS</i>	<A20><3*G10.0><I5>	File name for .TRS file related to truss elements
	<6*I5>	Nodes of truss elements
		Oblique supports
RELAX_ELEM	<A10>	Optional. Relaxation of end forces
BEAMS	<A5>	Relaxation in beams
ELEM	<A4><I10><6 or 14 reals>	Multiple lines possible, one for each relaxed beam
END_BEAMS	<A9>	
END_RELAX	<A9>	
“ PRECISION”		
“ LOADS”		
“ FUNCTION”		
“ NODELOAD”	<I10><6*G10.0>	Optional, multiple lines possible
“ DISTRBEAM”	<I10><NDIM*G10.0><I10>	Optional
“DISTRSH”	<I10><NDIM*G10.0><I10>	Optional
“DISTRSOLID”	<I10><NDIM*G10.0><I10>	Optional
“END_LOAD ”	line for end of section	
“ MASS”		Optional, only for dynamic analysis
“ M_NODE”		Optional
“ M_BEAM”		Optional
“ M_SHELL”		Optional
“END_MASS ”	line for end of section	
	Blank line for end of comments	
“ MATERIALS”		

	<A10><I5>	Material name, Number of temperatures
	<8*G10.0>	Material properties, multiple name-properties pairs possible
" TIME"		
" Timestep"	<G10.0><G10.0>	Multiple lines possible
" ENDTIME"		
" NOEPSTH" OR " EPSTH"		
"OUTPUT"		Optional
" TIMEPRINT"		
	<G10.0><G10.0>	Multiple lines possible
"END_TIMEPR"		
" PRINTDEPL"		Optional
"PRINTTMPRT"		Optional
" PRINTFHE"		Optional
"PRINTREACT"		Optional
" PRINTMN"		Optional
"PRNSIGMASL"		Optional
"PRINTVELAC"		Optional
"PRNSIGMASH"		Optional
"PRNNXSHELL"		Optional
"PRNEASHELL"		Optional
"PRNEISHELL"		Optional
"PRNSIGMABM"	<I><I>	Optional
" PRINTET"	<I><I>	Optional

" PRNEPSMBM	< >< >	Optional
"PRNEIBEAM"		optional
"PRNSIGMASL"		Optional
	Blank line for end of comments	

## 2.3 Material Properties

Material names are provided in the program by command CMAT(NM). the values of the parameters associated with this material are introduced in the PARACOLD vector. There is a maximum of eight values of PARACOLD(I,NM) available in the program, depending on the material name introduced in the CMAT(NM). Valid material names are:

- INSULATION, USER1, USER2, USER3, USER4, USER5, C\_GYPSUM, X\_GYPSUM and SFRM\_PROBA (these materials have only thermal properties; they do not carry any load),
- ELASTIC, BILIN, PARABCONC, RAMBOSGOOD, SILCO\_COLD and CALCO\_COLD (these materials have only 1D mechanical properties at room temperature),
- STEELEC3, STEELEC3EN, STEELEC3DC, PSTEELA16, STEELEC2, STEELEC2EN, STEEL\_WPB, STEELSL, STEC3PROBA, USER\_STEEL, CALCONCEC2, SILCONCEC2, LWCONC\_EN, SILCONC\_EN, CALCONC\_EN, SILCON\_ETC, CALCON\_ETC, SILHSC1\_EN, SILHSC2\_EN, SILHSC3\_EN, CALHSC1\_EN, CALHSC2\_EN, CALHSC3\_EN, SILHSC1ETC, SILHSC2ETC, SILHSC3ETC, CALHSC1ETC, CALHSC2ETC, CALHSC3ETC, SILCONC\_PR, CALCONC\_PR, WOODEC5, SLS1.4301, SLS1.4401, SLS1.4404, SLS1.4571, SLS1.4003, SLS1.4462, SLS1.4311, AL6061T6C, AL5083SUP, AL5083INF,

AL7020SUP and AL7020INF (these materials have thermal properties and 1D mechanical properties at elevated temperatures),

- STEELEC32D, SILCOETC2D, CALCOETC2D, LWCONC2D, SILCONC2D, CALCONC2D, STEELEC3PS, ELPLANESTR, PLSTRVML, BLPLSTRVM, BLPLSTRDP, VMRANK2D (these materials have 2D plane stress mechanical properties).
- SILCOETC3D, CALCOETC3D, STEELEC23D, STEELEC33D (these materials have thermal properties and 3D mechanical properties)

The stress-strain relationships in the load bearing materials are non-linear and are temperature dependent. In structures exposed to fire, the materials are subjected to initial strains ( $e_i$ ), thermal effects ( $e_{th}$ ) and stress related effects ( $e_s$ ). The stresses are, therefore, caused by the difference between the total strain ( $e_{total}$ ), obtained from the nodal displacements, and the initial and thermal strains.

## 2.4 Convergence Criteria

In order to converge to a solution, a tolerance value has to be specified in the program. SAFIR uses an iterative procedure to converge on the correct solution for each increment. The precision given in the data file is a small value that must be reached at different times in SAFIR calculations in order to have convergence. A good precision value is dependent on the type of structure that is being analyzed and information from preliminary runs. However, if the user does not know which to choose, a value of 0.001 can be used as a starting point (In case of a dynamic analysis, the default value of 0.0005 is recommended). After the first run, an examination in the output of the out-of-balance forces and increments of displacement during subsequent iterations can help the user to modify the corresponding precision value to obtain an acceptable solution.



## 3 DETAILED INPUT DATA AND FORMAT

### 3.1 Description and Format of the .IN file for Thermal Analysis

See “Users manual of Safir - Thermal.docx”, Section D.1.2

### 3.2 Description and Format of the eventual file describing the local HASEMI fire(s) for a thermal analysis

See “Users manual of Safir - Thermal.docx”, Section D.1.3

### 3.3 Description and Format of the .IN file for Structural Analysis

SERIES 1: Comments.

One line for each comment (can be 0 line).

SERIES 2:

One blank line to mark end of comments.

SERIES 3: Number of nodes.

One line.

"NNODE", NNODE

NNODE = Number of nodes of the structure.

SERIES 4: Number of axes.

One line.

"NDIM", NDIM

NDIM = Number of global axes, 2 for plane structures, 3 for 3-D structures.

SERIES 5: Degrees of freedom.

One line, first line in series.

"NDOFMAX", NDOFMAX

NDOFMAX = Maximum number of degrees of freedom per node.

if NDIM = 2 for truss elements, NDOFMAX  $\geq$  2

for solid elements, NDOFMAX  $\geq$  2

for beam elements, NDOFMAX  $\geq$  3

if NDIM = 3 for truss elements, NDOFMAX  $\geq$  3

for solid elements, NDOFMAX  $\geq$  3

for shell elements, NDOFMAX  $\geq$  6

for beam elements, NDOFMAX  $\geq$  7

SERIES 6 (optional): MATRIX SOLVER

1 card (optional)

SOLVER, CHOLESKY

Ø SOLVER

[A6]

command

Ø CHOLESKY

[A8]

command used to force SAFIR to solve the system of equations by the method of Cholesky with a storage of the matrix by a skyline method. This method is not as efficient as the default method of Pardiso based on a sparse matrix solver. It is thus not recommended to use this card. The possibility has been given as a safety measure because Pardiso has been introduced recently in the code; this card allows going back to the previous but outdated method in case any problem would appear with the new method.

If this card is used, the next card on `NCORES` cannot be used, because Cholesky systematically uses only one core of the computer.

1 card (optional)

`NCORES`, *ncores*

Ø `NCORES`

[A6]

command

Ø *ncores* [integer]

Number of cores of the CPU of the computer used by matrix solver. The default value is 1, in which case this card may be omitted. This card can be used to force SAFIR to use more than 1 core, if present on the computer. Recent experience has shown that using more than 1 core hardly reduces the time of the runs with the present version of Pardiso and this card can thus be omitted as a common practice. The possibility of using the card has nevertheless been given in order to allow users to perform their own test on their particular system, and in order to offer the possibility of working with more than 1 core in the future if new releases of Pardiso show to exploit several cores more efficiently.

This card cannot be used if the previous card of `CHOLESKY` has been used, because Cholesky systematically uses only one core of the computer.

SERIES 7: Loads.

One line, first line of three possible line series.

either

"STATIC ...\* "

if the structure or one part of it is submitted to the fire and a static analysis is required. This is the standard option.

or

"STATICCOLD ...\* "

if SAFIR is used to determine the ultimate load bearing capacity of a structure which is not submitted to the fire, i.e. at room temperature.

or

"DYNAMIC ...\* "

if the structure or one part of it is submitted to the fire and a dynamic analysis is required.

\* Static, staticcold and dynamic must be followed by the type of convergence procedure required during the structural analysis. The program can use a pure Newton-Raphson procedure ("PURE\_NR") or a modified Newton-Raphson procedure ("APPR\_NR"). "PURE\_NR" is recommended for structures made of beams, and "APPR\_NR" is recommended for structures made of shells.

Load number of vectors.

One line, second line of two line series.

'NLOAD', NLOAD

NLOAD = Number of load vectors. One load vector is made of different loads of different types that will vary with time according to the same function.

NEW SERIES Hydrostatic loads

One line

'HYDROST', NHYDROST

NHYDROST = number of hydrostatic loads. Hydrostatic loads may vary as a function of time during the calculation, depending on the level of the water table. They are not comprised in any load vector.

SERIES 8: Inclined supports.

One line.

'OBLIQUE', NOBLIQUE

NOBLIQUE = Number of inclined supports. Every node where a boundary condition is expressed in a local system of coordinates, instead of the global system of coordinates of the structure, is an oblique support, see Series 20.

A "0" must be typed if there is no oblique support.

**Note:**

The utilisation of oblique supports is only possible when the CHOLESKY solver is being used, see Series 5. It cannot be used with the PARDISO solver.

SERIES 9: Convergence strategy.

One line, first line of two line series, choice of two possible settings.

'COMEBACK', TimestepMIN

TimestepMIN = Minimum value for the time step in case of comeback only.

or

'NOCOMEBACK'

**Note:**

**For static analysis,**

If NOCOMEBACK is chosen, the simulation is stopped the first time the stiffness matrix is not positive definite.

If COMEBACK is chosen, each time the stiffness matrix is negative, time is reset at the last converged point and the simulation restarts from there with a time step divided by 2. The division of the time step goes on until the time step is smaller than TimestepMIN.

**For dynamic analysis,**

If NOCOMEBACK is chosen, the simulation is stopped the first time the left term of the equation is not positive.

If COMEBACK is chosen, each time this term is negative or the number of iterations necessary to obtain the convergence is greater than 3, time is reset at the last converged point and the simulation restarts from there with a smaller time step. The division of the time step goes on until the time step is smaller than TimestepMIN. If, for three simultaneous time steps, the convergence is

obtained in less than three iterations, the time step is multiplied by 2 (limited at the value of the initial time step).

SERIES 10:

Obsolete. No card needed here

SERIES 11: Number of materials.

One line.

'NMAT', NMAT

NMAT = Number of different materials.

**Note:**

If two materials have the same material law but different characteristics, it makes two different materials. e.g. S235 and S355 steel.

SERIES 12: Number of different elements.

One line, first line of multiple line series.

'ELEMENTS'

BEAM elements sub-series, present if beams are used in the structure.

One line

'BEAM', NBEAM, NGEOBEAM

NBEAM = Number of BEAM elements in the structure.

NGEOBEAM = Number of different groups of geometrical properties.

**Note:**

One group of geometrical properties comprises elements that have the same materials, the same cross section and the same temperature history. One .TEM file will be necessary to describe each of the NGEOBEAM groups.

One line.

'NG', NG

NG = Number of longitudinal points of integration in elements. Cannot be less than 2. Greater than 3 is not recommended.

One line.

'NFIBER', NFIBERBEAM

NFIBERBEAM = Number of longitudinal fibres in the beam elements (the maximum value for all the different groups of geometrical properties).

TRUSS elements subseries, present if truss elements are used in the structure

One line

'TRUSS', NTRUSS, NGEOTRUSS

NTRUSS = Number of TRUSS elements in the structure.

NGEOTRUSS = Number of different groups of geometrical

properties.

**Note:**

One group of geometrical properties comprised elements that had the same materials, the same cross sectional area and the same temperature history.

SHELL elements subseries, present if shell elements are used.

One line

'SHELL', NSHELL, NGEOSHELL

NSHELL = Number of SHELL elements in the structure.

NGEOSHELL = Number of different groups of geometrical

properties.

**Note:**

One group of geometrical properties comprised elements that had the same materials, the same thickness, the same reinforcing bars and the same temperature history.

One line

'NGTHICK', NGSHELLTHICK

NGSHELLTHICK = Number of points of integration on the thickness of the elements. Cannot be less than 2 and cannot be more than 10.

One line  
'NREBARS', NREBARS

NREBARS = Number of REBAR layers in the shell elements.

SOLID elements subseries, present if solid elements are used

One line  
'SOLID', NSOLID  
NSOLID = number of SOLID elements in the structure

One line.  
'NG', NG  
NG = Number of integration points. Valid entries are 1, 2 and 3.

SPRING elements subseries, present if spring elements are used

One line  
'SPRING', NSPRING  
NSPRING = number of SPRING elements in the structure

Last line of series.

"END\_ELEM"

SERIES 13: The nodes.

One line, first line of multiple line series.

<A10>,<A10>]

'NODES'

or

'NODES\_CYL'

'NODES\_CYL' is used if the cylindrical system of co-ordinate is chosen instead of the Cartesian system for the introduction of the co-ordinates of the nodes. Cylindrical input are transformed for the internal solution process by

$(r,q) \Rightarrow X = r \cos(q), Y = r \sin(q)$ . if NDIM = 2

$(r,q,Z) \Rightarrow X = r \cos(q), Y = r \sin(q), Z$ , if NDIM = 3

**Note:**

q is in degrees.

The transformation is made after all the nodes have been read and generated. CYLINDRIC is omitted if the nodes are directly input in the Cartesian system of co-ordinates.

Nodes.

One line added for each node.

'NODE', NNO, RCOORD(1,NNO), ..., RCOORD(NDIM,NNO)

NNO = Number of the specific node.

RCOORD(1,NNO) = First global coordinate of the node NNO.

...

...

RCOORD(NDIM,NNO) = Last global coordinate of node  
NNO of NDIM global axis.

or 'GNODE', NNO, RCOORD(1,NNO), ..., RCOORD(NDIM,NNO)

NNO = Number of the specific node.

RCOORD(1,NNO) = First global coordinate of the node NNO.

...

...

RCOORD(NDIM,NNO) = Last global coordinate of node  
NNO of NDIM global axis.

This command is used to generate equidistant nodes  
between the previously defined node and the current  
node NNO.

or 'REPEAT', NNO, DELTAC(1), ..., DELTAC(NDIM), KGENE

NNO = Number of nodes to be repeated.

DELTAC(1) = Increment on the first coordinate.

...

...  
DELTA(NDIM) = Increment on the coordinate NDIM.  
KGENE = Number of times that this command has to be repeated.

SERIES 14: Supports and imposed displacements.  
One line, first line of possible multiple line series.  
'FIXATIONS'

Supports and imposed displacements fixed blocks.

One line for each node where solution follows a defined function of time and the reaction must be calculated.

'BLOCK', NNO, CBLOCK(1,NNO), ..., CBLOCK(NDOFMAX,NNO)

NNO = Number of the specific node where the solution must not be calculated.

CBLOCK(1,NNO) = Function describing displacement for first D.o.F. at this node with respect to time. Type NO if the displacement is not prescribed for this DoF.

CBLOCK(2,NNO) = Function describing displacement for second D.o.F. at this node with respect to time Type NO if the displacement is not prescribed for this DoF.

...

CBLOCK(NDOFMAX,NNO) = Function describing displacement for last D.o.F. at this node with respect to time. Type NO if the displacement is not prescribed for this DoF.

**Note:**

For each degrees of freedom NDL, from 1 to NDOFMAX, CBLOCK(NDL,NNO) is either 'NO' if the displacement is not imposed at this D.o.F. or the name of the function describing the evolution of the displacement at this node with respect to time.

'F0' is a common function, used to model a fixed support.

**Note:**

The next lines with the SAME commands can come only after all BLOCK commands have been entered.

Supports and imposed displacements slave nodes.

One line added for each slave node.

'SAME', NNO1, NNO2, CTRAV(1), ..., CTRAV(NDOFMAX)

NNO1 = Number of the specific slave node.

NNO2 = Number of the master node.

CTRAV(1) = 'YES' If the solution is the same as at node NNO2  
and as at node NNO1 for the D.o.F. 1, = 'NO' otherwise

...

CTRAV(NDOFMAX) = 'YES' If the solution is the same as at node  
NNO2 and as at node NNO1 for the D.o.F. NDOFMAX, = 'NO'  
otherwise.

or one line added for repeating series of slave node nodes

'REPEAT', NUMBER, INCR, CTRAV(1), ..., CTRAV(NDOFMAX)

NUMBER = Number of times that the preceding SAME  
command must be repeated.

INCR = Increment on NNO1 and NNO2.

CTRAV(1) = 'YES' If the solution is the same as at node  
NNO2 as at node NNO1 for the D.o.F. 1, = 'NO' If  
there is no master-slave relation for this D.o.F.

...

CTRAV(NDOFMAX) = 'YES' If the solution is the same as at  
NNO2 as at NNO1 for the last D.o.F. = 'NO' If there is  
no master-slave relation for this D.o.F.

Or one line to create master-slave relationships  
between all nodes with same coordinates

'SAMEALL', CTRAV(1), ..., CTRAV(NDOFMAX)

All the nodes of the structure that have the same coordinates (with a  
precision of 0.1 mm) will automatically be attributed a master-slave  
relationship.

CTRAV(1) = 'YES' If the solution is the same for the D.o.F. 1, =  
'NO' otherwise

...

CTRAV(NDOFMAX) = 'YES' If the solution is the same for the  
D.o.F. NDOFMAX, = 'NO' otherwise.

**Note:**

A node cannot be defined as slave node in a relationship and master node in another relationship.

Therefore, if a given node is used in two or more relationships of slave-master, it must be defined whether as the master in all these relationships, or as the slave in all these relationships.

**Example:**

Consider that nodes 1, 2 and 3, which have 7 DoF, coincide and that these three nodes must have the same displacement in the z direction (in global axes). The following relationships are valid:

SAME 2 1 NO NO YES NO NO NO NO  
SAME 3 1 NO NO YES NO NO NO NO

The following relationships are **not valid**:

SAME 2 1 NO NO YES NO NO NO NO  
SAME 1 3 NO NO YES NO NO NO NO

Last line, indicating that the series is finished  
'END\_FIX'

SERIES 15: BEAM elements.

**Note:**

This series is skipped if no BEAM element is present in the structure.

One line, first line of possible multiple line series.  
'NODOFBEAM'

Beam elements file name sub-series. One sub-series for each type of element.  
One line, first line of sub-series.

<A20>

'CFILENAME'

CFILENAME = Full name of the file where the information on this section type can be found. Usually the extension is .TEM. File name is left justified.

**Note:**

The name of the .TEM files that describe the sections heated by the HASEMI fire is, for each section type, the name of ONE of the relevant .TEM file. For example, "b0017\_2.tem".

The information about the torsion properties has to be present only in this file, not in the other .TEM files of the same section type that describe the temperature at the other points of integration.

As a consequence, all the beam elements of one section type have the same torsion stiffness.

Beam elements sub-series material translation.

One line added for each different material used in the section.

'TRANSLATE', MATL, MATG

MATL = Local number of this material in this section type.

MATG = Global number of this material in the structure.

**Note:**

MATL starts from 1 for the first material in this section type. The second line is for the 2nd local material, etc.

Those lines are necessary because of the strategy used for the data files. One structure can be made of several BEAM section types, each of them being described in one .TEM file. In each of those .TEM files, the different materials are given numbers starting from 1. It is necessary to indicate at the level of the structure, which global material number corresponds to the numbers given in the .TEM files.

Beam element sub series last line.

One line to mark end of sub-series.

'END\_TRANS'

Beam elements list (in increasing order, from 1 to NBEAM).

'ELEM', NE, NODOFBEAM(1,NE), ..., NODOFBEAM(4,NE),  
ITYPEBEAM(NE)

NE = Number of this element.

NODOFBEAM(1,NE) = First end node of this element.

NODOFBEAM(3,NE) = Third (i.e. central) node of this element.

NODOFBEAM(2,NE) = Second end node of this element.

NODOFBEAM(4,NE) = 4<sup>th</sup> node of this element (present  
only if NDIM = 3).

ITYPEBEAM(NE) = The section type of this element.

or

'GELEM', NE, NODOFBEAM(1,NE), ..., NODOFBEAM(4,NE),  
ITYPEBEAM(NE), KGENE

KGENE allows the generation from the previously defined  
element up to this one. KGENE gives the increment  
on the first 3 nodes.

or

'REPEAT', NE, Nincr123, Nincr4, NincrType, Ntimes

NE = The NE previously defined elements will be repeated

NINCR123: increment on the nodes 1, 2 and 3

NINCR4: increment on the node 4 (present only if NDIM = 3).

NINCRTYPE: increment on the type of the element.

NTIMES: how many times these NE elements will be repeated.

Example:

The following sequence

```
ELEM 1 1 1 2 3 108 1
GELEM 8 15 16 17 108 1 2
ELEM 9 18 19 20 108 1
GELEM 16 32 33 34 108 1 2
ELEM 17 35 36 37 108 1
GELEM 24 49 50 51 108 1 2
ELEM 25 52 53 54 108 1
GELEM 32 66 67 68 108 1 2
ELEM 33 69 70 71 108 1
GELEM 40 83 84 85 108 1 2
```

can be replaced by the following one

```
ELEM 1 1 2 3 108 1
GELEM 8 15 16 17 108 1 2
REPEAT 8 17 0 0 4
```

to generate

ELEM.	NODE 1	NODE 3	NODE 2	NODE 4	TYPE	LENGTH
1	1	2	3	108	1	0.1288E+01
2	3	4	5	108	1	0.1288E+01
3	5	6	7	108	1	0.1288E+01
4	7	8	9	108	1	0.1288E+01
5	9	10	11	108	1	0.1200E+01
6	11	12	13	108	1	0.1200E+01
7	13	14	15	108	1	0.1200E+01
8	15	16	17	108	1	0.1200E+01
9	18	19	20	108	1	0.1288E+01
10	20	21	22	108	1	0.1288E+01
11	22	23	24	108	1	0.1288E+01
12	24	25	26	108	1	0.1288E+01
13	26	27	28	108	1	0.1200E+01
14	28	29	30	108	1	0.1200E+01
15	30	31	32	108	1	0.1200E+01
16	32	33	34	108	1	0.1200E+01
17	35	36	37	108	1	0.1288E+01
18	37	38	39	108	1	0.1288E+01
19	39	40	41	108	1	0.1288E+01
20	41	42	43	108	1	0.1288E+01

21	43	44	45	108	1	0.1200E+01
22	45	46	47	108	1	0.1200E+01
23	47	48	49	108	1	0.1200E+01
24	49	50	51	108	1	0.1200E+01
25	52	53	54	108	1	0.1288E+01
26	54	55	56	108	1	0.1288E+01
27	56	57	58	108	1	0.1288E+01
28	58	59	60	108	1	0.1288E+01
29	60	61	62	108	1	0.1200E+01
30	62	63	64	108	1	0.1200E+01
31	64	65	66	108	1	0.1200E+01
32	66	67	68	108	1	0.1200E+01
33	69	70	71	108	1	0.1288E+01
34	71	72	73	108	1	0.1288E+01
35	73	74	75	108	1	0.1288E+01
36	75	76	77	108	1	0.1288E+01
37	77	78	79	108	1	0.1200E+01
38	79	80	81	108	1	0.1200E+01
39	81	82	83	108	1	0.1200E+01
40	83	84	85	108	1	0.1200E+01

SERIES 16: SOLID elements.

**Note:**

This series is skipped if no SOLID element is present in the structure.

One line, first line of possible multiple line series.

<A10>

'NODOFSOLID'

Solid elements file list. One sub-series for each type of element.

One line, first line of sub-series.

<A20>

left justified

'CFILENAME'

CFILENAME = Full name of the file where the information on the temperature distribution in the structure can be found. Usually the extension is .OUT; this file is the result of the 3D thermal analysis.

Solid elements list.

One line for each solid element.

<A10>,<9\*15>

'ELEM', NSL, N1, N2, N3, N4, N5, N6, N7, N8, NMAT, RES1,  
RES2, RES3

NSL = Number of the element.

N1 = Node 1

N2 = Node 2

N3 = Node 3

N4 = Node 4

N5 = Node 5

N6 = Node 6

N7 = Node 7

N8 = Node 8

NMAT = Number of the material.

RES1 = Residual stress in the element in the direction of the  
global axis 1.

RES2 = Residual stress in the element in the direction of the  
global axis 2.

RES3 = Residual stress in the element in the direction of the  
global axis 3.

or

<A10>,<I5>,<I5>,<30b>,<I5>

' REPEAT', NSL, N1, ' ', NUMBER

NSL = Number of elements to be repeated.

N1 = Node increment.

NUMBER = Number of times that the NSL elements have to be  
repeated.

SERIES 17: SHELL elements.

**Note:**

This series is skipped if no SHELL element is present in the structure.

One line, first line of possible multiple line series.

<A10>

'NODOSHELL'

Shell elements file list.

One line, first part of shell element sub-series, one sub-series for each element type.

<A20> left justified

'CFILENAME'

CFILENAME = File name where the information concerning this section type is read.

**Note:**

The name of the .TSH files that describe the sections heated by the HASEMI fire is, for each section type, the name of ONE of the relevant .TSH file. For example, "s0156\_3.tsh".

The information about the re-bar layers has to be present only in this file, not in the other .TSH files of the same section type that describe the temperature at the other points of integration.

As a consequence, all the shell elements of one section type have the same re-bars.

Shell element material translation.

One line for each different material in section, second part of shell element sub-series.

<A10>,<I5>,<I5>

'TRANSLATE', N1, N2

N1 = Local number of this material in this section type.

N2 = Global number of this material in the structure.

Shell element series end of translation.

One line.

<A10>

'ENDTRANSLA'

Shell element list.

One line for each shell element.

<A10>,<9\*I5>

'ELEM', NSH, N1, N2, N3, N4, ITYPESHELL(NSH), KGENE

NSH = Number of the element.

N1 = Node 1

N2 = Node 2

N3 = Node 3

N4 = Node 4

ITYPESHELL(NSH) = Type of geometrical section.

KGENE = Automatic generation on the element number.

or

<A10>,<I5>,<I5>,<30b>,<I5>

‘ REPEAT’, NSH, N1, ‘ , NUMBER

NSH = Number of elements to be repeated.

N1 = Node increment.

NUMBER = Number of times that the NSH elements have to be repeated.

SERIES 18: TRUSS elements.

**Note:**

This series is skipped if no TRUSS element is present in the structure.

One line, first line of possible multiple line series.

‘NODOFTRUSS’

Truss elements files.

One line for each different truss section type used.

‘CFILENAME’, GEOTRUSS(1,NGT), GEOTRUSS(2,NGT),  
IMATTRUSS(NGT)

CFILENAME = Name of the file where the temperatures concerning this section types are read. Left justified.

GEOTRUSS(1,NGT) = Cross sectional area of this section type.

GEOTRUSS(2,NGT) = Residual stress of this section type.

IMATTRUSS(NGT) = Global Number of the material in this section

type.

**Note:**

If cfilename (NGT) is left blank, then:

- this must be the case for all the section types
- there is only one element in each NGT
- the elements must be linked to nodes which belong to solid elements
- the temperature of each truss element is the average of the temperature of its 2 nodes, calculated with solid elements.

Truss elements list.

One line for each truss element.

```
'ELEM', NTR, NODOFTRUSS(1,NTR),  
        NODOFTRUSS(2,NTR), IGEOTRUSS(NTR),  
        KGENE
```

NTR = Number of the element.

NODOFTRUSS(1,NTR) = First node of this element.

NODOFTRUSS(2,NTR) = Second node of this element.

IGEOTRUSS(NTR) = Number of the section type for this element.

KGENE = Allows for automatic generation.

SERIES 19: SPRING elements.

**Note:**

This series is skipped if no SPRING element is present in the structure.

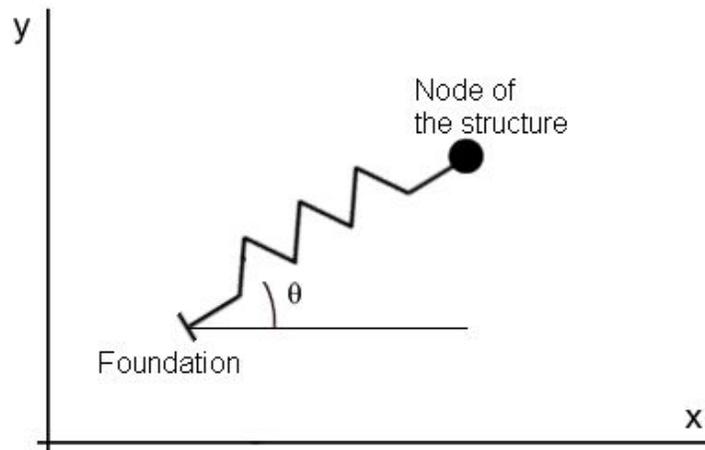
One line, first line of possible multiple line series.

```
'NDFSPRING'
```

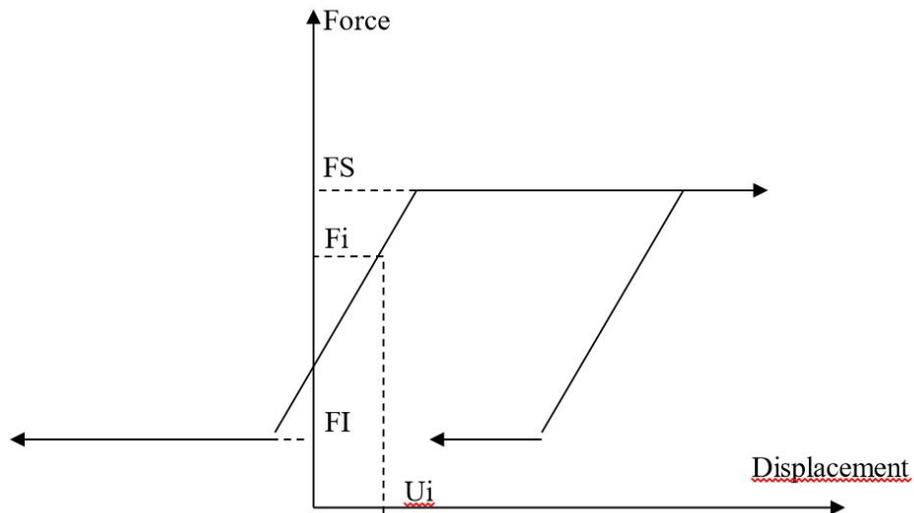
Each element is first defined by the node of the structure on which it is applied.

The positive direction of the spring element is always defined from the foundation to the structure, which means that, in the output, a positive force in the spring, corresponding to tension, will be pulling on the structure, whereas a negative force will push on the structure.

In the case of 2D models, the direction of the element is defined by the angle at the foundation between the global axis X and the element, in counter clockwise direction, as shown on the figure below.



In the case of 3D models, the direction of the element is defined by a 3D vector that has the direction from the foundation to the node of the structure. This vector does not have to be normalised.



The behaviour of the spring is shown on the previous Figure.  
 The force in the spring is comprised between two limits: the upper limit  $FS$  and the lower limit  $FI$ .  
 The slope of the elastic part is the stiffness  $K$ .  
 The state of the spring at time  $t=0$  is defined by the initial displacement  $U_i$  and initial force  $F_i$ .  
 The behaviour in unloading is plastic.

All forces from the F-u diagram will be multiplied by the area of influence A. As a consequence, the stiffness is also multiplied by A.

Note: in contradiction with the sign convention that is used in the output, the values of FS, FI and Fi (as well as K) are positive when the spring pushes on the structure. This is by analogy with active and passive pressure when the spring is used to model a soil beside a wall or a foundation underneath a beam.

One line for each SPRING element

For 2D structures

'ELEM', NSPR, NNODE, THETA, FS, FI, K, A, Ui, Fi

NSPR        Number of the element.

NNODE      Node where this element is attached.

THETA      Angle between X axis and this element  
[degrees].

FS          Superior limit of the force.

FI          Inferior limit of the force.

K          Stiffness of the element.

A          Area of influence.

Ui          Displacement in the configuration of reference  
(time t = 0).

Fi          Force in the configuration of reference (time t =  
0).

For 3D structures

'ELEM', NSPR, NNODE, CX, CY, CZ, FS, FI, K, A, Ui, Fi

NSPR        Number of the element.

NNODE      Node where this element is attached.

CX          Component X of the vector that defines the  
direction.

CY          Component Y of the vector that defines the  
direction.

CZ          Component Z of the vector that defines the  
direction.

FS          Superior limit of the force.

FI          Inferior limit of the force.

	K	Stiffness of the element.
	A	Area of influence.
(time $t = 0$ ).	$U_i$	Displacement in the configuration of reference
	$F_i$	Force in the configuration of reference (time $t = 0$ ).

SERIES 19-2: Oblique supports (only possible with the CHOLESKY solver)

One line for each oblique support.

For 2D structures

'INCLIN',  $N_i$ ,  $N_j$

$N_i$  is the node where a boundary condition is expressed in a local system of coordinates.

$N_j$  is another nodes of the structure.

$N_i$  and  $N_j$  define the direction in which the node  $N_i$  can move. It cannot move perpendicularly to this direction.

For 3D structures

Boundary condition applied to the displacements

'INCLIN',  $N_i$ ,  $N_j$ ,  $N_k$

$N_i$  is the node where a boundary condition is expressed in a local system of coordinates.

$N_j$ ,  $N_k$  are 2 other nodes of the structure.

$N_i$ ,  $N_j$  and  $N_k$  define the plane in which the node  $N_i$  can move. It cannot move out of this plane.

Boundary condition applied to the rotations

'INCLIN',  $-N_i$ ,  $N_j$ ,  $N_k$

$N_i$  is the node where a boundary condition is expressed in a local system of coordinates.

$N_j$ ,  $N_k$  are 2 other nodes of the structure.

Ni, Nj and Nk define the plane in which the node Ni can have rotations. It cannot rotate along an axis perpendicular to this plane.

One line to indicate that this is the end of the Series 20-0  
<A10>  
'END\_INCLIN'

### SERIES 19-3 Relaxation

This series is optional.

It is used to relax the internal force at the end of some elements.

The internal force F at the designated degree of freedom is computed according to the following equation:

$$F = K (u_s - u_i)$$

where K is a stiffness;  
u<sub>s</sub> is the displacement of the node of the structure, in the local system of coordinates of the beam;  
u<sub>i</sub> is the displacement of the internal node at the end of the element, in the local system of coordinates of the beam;

One line  
'RELAX\_ELEM'

One line  
'BEAMS'

This line indicates that at least one BEAM finite element has one or several degrees of freedom that are relaxed.

One line for each BEAM element that has one or several relaxed degrees of freedom.

'ELEM', NBM, K(1), K(2), .... K(Idimk-1)

NBM = Number of the BEAM element

K(1) = stiffness for the relaxation at the first degree of freedom of the beam (translation along local axis x of the beam at node 1).

If the value is 0, there is no spring between the beam element and the node of the structure. The relaxation is total and the force at the end of the beam element is equal to 0.

If the value is negative, there is no relaxation. The displacement at the end of the beam is equal to the displacement of the structure.

$K(2)$  = stiffness at the first degree of freedom of the beam.

...

$K(idimk-1)$  = stiffness at the last rotational degree of freedom of node 2

Notes:

- 1)  $Idimk$  is the number of degrees of freedom of the beam, 7 for 2D beams and 15 for 3D beams.
- 2) The degree of freedom of the central node cannot be relaxed.

One line  
'END\_BEAMS'

One line  
'END\_RELAX'

SERIES 20: Precision.

One line.

'PRECISION', PRECISION

PRECISION = Small value that must be reached for convergence.

NEW SERIES: Limiting displacement

This series exists only if a static or a dynamic calculation is being performed

One line

'MAX\_DISPL', MAX\_DISPL

MAX\_DISPL = maximum value admitted for the displacements. If this value is exceeded for an incremental displacement during iteration, a COMEBACK is performed, if it has been foreseen, or the calculation is stopped. If this value is exceeded after convergence, the calculation is stopped.

Note: this series can be omitted (which ensures compatibility with SAFIR versions older than 2016b). In that case, MAX\_DISPL is given the value of 999.

The SERIES 21 is repeated NLOAD times

SERIES 21: Loading.

One line.

'LOADS'

Loading function.

One line

'FUNCTION', CFORCE(NLO)

CFORCE(NLO) = Function showing how the load vector NLO varies as a function of time, see § 1.5.2.

Loading on nodes

One line added for each point load directed at a node.

'NODELOAD', NNO, LOAD(1), LOAD(2), ..., LOAD(NDOF)

NNO = Number of the node where loads are applied.

LOAD(1) = Load at degrees of freedom 1.

LOAD(2) = Load at degrees of freedom 2.

...

LOAD(NNDL) = Load at degrees of freedom NDOF

Note: iloc = minimum(NDOF,6)

Loading on beam elements.

Uniformly distributed loads

The direction of these loads is according to the global system of coordinates and they are measured per meter in the direction of the local axis x of the beam finite element. For example, the dead

weight of a prismatic column that extends along the global axis Z can be described by a DISTRBEAM load in the direction of the global axis Z.

One line added for each element with a uniformly distributed load applied.

'DISTRBEAM', NBM, TRAV(1), TRAV(2), ..., TRAV(NDIM)

NBM = Number of the specific BEAM under a distributed load.

TRAV(1) = Uniformly distributed load in the direction of the global axis 1.

TRAV(2) = Uniformly distributed load in the direction of the global axis 2.

...

TRAV(NDIM) = Uniformly distributed load in the direction of the final global axis.

or

'GDISTRBEAM', NBM, TRAV(1), TRAV(2), ..., TRAV(NDIM),

KGENE

NBM = Number of the specific BEAM under a distributed load.

TRAV(1) = Uniformly distributed load in the direction of the global axis 1.

...

TRAV(NDIM) = Uniformly distributed load in the direction of the final global axis.

KGENE , distributed loads are generated between the previously defined element and the present element with a step on the element numbers of KGENE

Trapezoidal distributed loads in local axes

The direction of these loads is according to the local system of coordinates of the beam and they are measured per meter in the direction of the local axis x of the beam. Such load can be used, for example, to impose a wind pressure that varies in a trapezoidal manner along an inclined roof.

One line added for each element with a distributed load applied.

'TRAPLOCBM', NBM, TRAV(1), TRAV(2), ..., TRAV(2\*NDIM)

NBM = Number of the BEAM under a trapezoidal distributed load.

TRAV(1): distributed load in the direction of the local axis 1 at node 1 of the element.

TRAV(2): distributed load in the direction of the local axis 2 at node 1.

...

TRAV(NDIM): distributed load in the direction of the final local axis at node 1.

TRAV(NDIM+1): distributed load in the direction of the local axis 1 at node 2 of the element.

TRAV(NDIM+2): distributed load in the direction of the local axis 2 at node 2.

...

TRAV(2\*NDIM): distributed load in the direction of the final local axis at node 2.

or

'GTRAPLOCBM', NBM1,NBM2, TRAV(1), TRAV(2), ..., TRAV(2\*NDIM), KGENE

NBM1: Number of the first BEAM under a trapezoidal distributed load.

NBM2: Number of the last BEAM under a trapezoidal distributed load.

TRAV(1): distributed load in the direction of the local axis 1 at node 1 of the element NBM1.

TRAV(2): distributed load in the direction of the local axis 2 at node 1 of the element NBM1..

...

TRAV(NDIM): distributed load in the direction of the final local axis at node 1 of the element NBM1..

TRAV(NDIM+1): distributed load in the direction of the local axis 1 at node 2 of the element NBM2.

TRAV(NDIM+2): distributed load in the direction of the local axis 2 at node 2 of the element NBM2.

...

TRAV(2\*NDIM): distributed load in the direction of the final local axis at node 2 of the element NBM2..

KGENE , distributed loads are generated with a step on the element numbers of KGENE. The value of the trapezoidal loads at the intermediate nodes are calculated by linear interpolation based on the length of the elements

#### Trapezoidal distributed loads in global axes

The direction of these loads is according to the global system of coordinates and they are measured per meter in the direction of the local axis x of the beam. Such load can be used, for example, to impose the dead weight of a tapered member.

One line added for each element with a distributed load applied.

'TRAPGLOBM', NBM, TRAV(1), TRAV(2), ..., TRAV(2\*NDIM)

or

'GTRAPGLOBM', NBM1,NBM2, TRAV(1), TRAV(2), ...,  
TRAV(2\*NDIM), KGENE

The format of these commands is exactly the same as the format of TRAPLOCBM and GTRAPLOCBM, except that the distributed loads are here described in the direction of the global system of coordinates.

Loading on shell elements.

One line added for each element with a distributed load applied.

'DISTRSH', NSH, TRAV(1), TRAV(2), TRAV(3)

NSH Number of the specific SHELL element under a distributed load.

TRAV(1) Uniformly distributed load in the direction of the global axis 1.

TRAV(2) Uniformly distributed load in the direction of the global axis 2.

TRAV(3) Uniformly distributed load in the direction of the global axis 3.

or

'GDISTRSH', NSH, TRAV(1), TRAV(2), TRAV(3), KGENE

NSH Number of the specific SHELL element under a distributed load.

TRAV(1) Uniformly distributed load in the direction of the global axis 1.

TRAV(2) Uniformly distributed load in the direction of the global axis 2.

TRAV(3) Uniformly distributed load in the direction of the global axis 3.

KGENE Increment on the element number

Loading end of series.

One line

'END\_LOAD'

NEW SERIES: hydrostatic loads

This series is repeated NHYDROST time

One line

'WATERTABLE', CWATERTABLE

CWATERTABLE = chain of (maximum 10) characters, function describing the level (in meters) of the water table

One line

'SPECWEIGHT', GAMMA

GAMMA = specific weight of the fluid, in  $m(N/m^3)$ .

Note: The first meter in the dimension of this value represents the horizontal distance between adjacent beam elements. For example, if 2D frames are representing frames that are, in fact, 3 meters away in the direction perpendicular to the plane of the frame, the specific weight of water to be considered here is not  $10.000 N/m^3$ , but  $30.000 m(N/m^3)$

If the specific weight is positive, the load on each element is in the direction of the local axis y of the element. It is possible to enter a negative value of the specific weight to change the direction of the load.

Multiple lines, giving the list of the beam elements loaded by this hydrostatic load

One line

'HYDROBM', NBM

NBM = number of the element loaded by this hydrostatic load.

or

One line

'GHYDROBM', NBM, KGENE

The elements from the previously defined element to the element NBM, with an increment of KGENE, are loaded by this hydrostatic load.

One line

'END\_HYDRO'

Note: A beam element loaded by a hydrostatic load will be loaded by a TRAPLOCBM load. The distributed load at each end node of the element is given by

$$P = \text{GAMMA} \times \text{DEPTH}$$

where DEPTH is the difference between the value of CWATERTABLE and the coordinate NDIM of the node in the deformed configuration. It is thus assumed that gravity is in the direction of  $-Y$  for 2D structures and  $-Z$  for 3D structures. If the value of DEPTH at a given node is negative, it is replaced by 0.

The pressure will be applied on the deformed shape of the structure.

SERIES 22: Mass characteristic.

Notes:

- 1) This series is present ONLY IF DYNAMIC HAS BEEN CHOSEN
- 2) In SAFIR, masses and forces are totally independent. The masses introduced produce no force and the forces are not linked to any mass. As a consequence, if a force of X Newton is produced by gravity, a mass of X/10 kg must normally be also introduced in the data; if a force is produced by wind, no mass has to be introduced.

One line, first line of possible multiple line series.

'MASS'

Concentrated mass on nodes.

One line added for each concentrated mass linked to a node.

'M\_NODE', NNO, MASS(1), MASS(2), ..., MASS(NDOF)

NNO = Number of the node where the mass are applied.

MASS(1) = Mass linked to degree of freedom 1.

MASS(2) = Mass linked to degree of freedom 2.

...

MASS(NDOF) = Mass linked to degree of freedom NDOF.

Notes:

- 1) A mass linked to a displacement is in kg. A mass linked to a rotation is in  $\text{kgm}^2$ .
- 2) Usually, a concentrated mass linked to a displacement is active in all directions. Only in some particular cases can a mass be inactive in a particular direction (for example, a ball laying on a horizontal surface might be active in the direction perpendicular to this surface, and not active in the directions parallel to this surface)

Distributed mass on beam elements.

One line added for each beam element with a distributed mass applied.

'M\_BEAM', NBM, TRAV(1), TRAV(2)

NBM = Number of the specific BEAM under a distributed mass.

TRAV(1) = Uniformly distributed mass applied on the beam element (kg/m).

TRAV(2) = Rotational inertia around the longitudinal axis of the element

with TRAV(2) only present if a 3D analysis is made.

or

'GM\_BEAM', NBM, TRAV(1), TRAV(2), KGENE

NBM = Number of the specific BEAM under a distributed mass.

TRAV(1) = Uniformly distributed mass applied on the beam element (kg/m).

TRAV(2) = Rotational inertia around the longitudinal axis of the element

(TRAV(2) only present if a 3D analysis is made)

KGENE = Increment on the element number

(Distributed mass are generated between the previously defined element and the present element)

Distributed mass on shell elements.

One line added for each shell element with a distributed mass applied.

'M\_SHELL', NSH, TRAV(1)

NSH Number of the specific SHELL element under a distributed mass.

TRAV(1) Uniformly distributed mass on the shell element (kg/m<sup>2</sup>).

or

'GM\_SHELL', NSH, TRAV(1), KGENE

NSH Number of the specific SHELL element under a distributed load.

TRAV(1) Uniformly distributed mass on the shell element (kg/m<sup>2</sup>).

KGENE Increment on the element number

Mass end of series.

One, last line of series.

'END\_MASS'



SERIES 23: Material description.

One line, first line of possible multiple line series.

MATERIALS

Material description sub-series. One sub-series entered for each NMAT material type

One line, first line of two line material sub-series.

CMAT

CMAT Name of the material

**Valid material names are:**

INSULATION, USER1, USER2, USER3, USER4, USER5,  
X\_GYPSUM, C\_GYPSUM, SFRM\_PROBA

ELASTIC, CALCON\_ETC, SILCON\_ETC, CALCONC\_EN,  
SILCONC\_EN, SILHSC1ETC, SILHSC2ETC, SILHSC3ETC,  
CALHSC1ETC, CALHSC2ETC, CALHSC3ETC, SILHSC1\_EN,  
SILHSC2\_EN, SILHSC3\_EN, CALHSC1\_EN, CALHSC2\_EN,  
CALHSC3\_EN, CALCONC\_PR, SILCONC\_PR, CALCONCEC2,  
SILCONCEC2, LWCONC\_EN, PARABCONC, CALCO\_COLD,  
SILCO\_COLD, STEELEC3, STEELEC3EN, STEELEC3DC,  
STEELSL, STEELEC2, STEELEC2EN, SLS1.4301, SLS1.4401,  
SLS1.4404, SLS1.4571, SLS1.4003, SLS1.4462,  
SLS1.4311, BILIN, RAMBOSGOOD, WOODEC5, AL6061T6C,  
AL5083INF, AL5083SUP, AL7020INF, AL7020SUP,  
USER\_STEEL, PSTEELA16, STEEL\_WPB, STEC3PROBA

ELPLANESTR, PLSTRVML, STEELEC32D, SILCOETC2D,  
CALCOETC2D, SILCONC2D, CALCONC2D, LWCONC2D,  
STEELEC3PS, VMRANK2D, BLPLSTRVM, BLPLSTRDP

SILCOETC3D, CALCOETC3D, STEELEC23D, STEELEC33D

Material description sub-series parameters.

One line, second line of two line material sub-series.



## INSULATION MATERIAL TYPES

If CMAT = INSULATION, USER1, USER2, USER3, USER4, USER5, X\_GYPSUM, C\_GYPSUM, SFRM\_PROBA, no parameter is necessary because this material does not carry any stress. In this case, the second line is a blank line.

## UNIAXIAL MATERIAL TYPES

If CMAT = ELASTIC (this material is valid only at 20°C.)

PARACOLD(1,NM) Young's modulus.

PARACOLD(2,NM) Poisson ratio.

If CMAT = BILIN (this material is valid only at 20°C.)

PARACOLD(1,NM) Young's modulus.

PARACOLD(2,NM) Poisson ratio.

PARACOLD(3,NM) Yield strength.

PARACOLD(4,NM) Slope of the hardening branch.

If CMAT = RAMBOSGOOD (this material is valid only at 20°C.)

PARACOLD(1,NM) E, Young's modulus.

PARACOLD(1,NM) Poisson ratio

PARACOLD(3,NM)  $l_p$ , the limit of proportionality.

PARACOLD(4,NM) n, exponent of the law.

PARACOLD(5,NM) K, factor of the law.

$\epsilon = \sigma/E$  for  $\sigma \leq l_p$

$\epsilon = (\sigma/E) + ((\sigma-l_p)/K)^n$  for  $\sigma > l_p$

If CMAT = CALCON\_ETC, SILCON\_ETC

PARACOLD(2,NM) Poisson ratio.

PARACOLD(3,NM) Compressive strength

PARACOLD(4,NM) Tensile strength ( $\geq 0$ )

The CALCON\_ETC, SILCON\_ETC materials take into account transient creep strain explicitly. The models are described in:

Gernay T., Franssen J.M. (2012). "A formulation of the Eurocode 2 concrete model at elevated temperature that includes an explicit term for transient creep". *Fire Safety Journal*, 51, 1-9.

If CMAT = CALCONC\_EN, SILCONC\_EN, LWCONC\_EN

PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Compressive strength  
PARACOLD(4,NM) Tensile strength ( $\geq 0$ )

If CMAT = CALCONCEC2, SILCONCEC2

PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Compressive strength  
PARACOLD(4,NM) Tensile strength ( $\geq 0$ )  
PARACOLD(5,NM)

< 0 if peak stress strain  $e_{c1}$  = minimum value (stiffer)

= 0 if peak stress strain  $e_{c1}$  = recommended value

> 0 if peak stress strain  $e_{c1}$  = maximum value (more ductile)

If CMAT = SILHSC1ETC, SILHSC2ETC, SILHSC3ETC,  
CALHSC1ETC, CALHSC2ETC, CALHSC3ETC

PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Compressive strength  
PARACOLD(4,NM) Tensile strength ( $\geq 0$ )

If CMAT = SILHSC1\_EN, SILHSC2\_EN, SILHSC3\_EN,  
CALHSC1\_EN, CALHSC2\_EN, CALHSC3\_EN

PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Compressive strength  
PARACOLD(4,NM) Tensile strength ( $\geq 0$ )

If CMAT = CALCONC\_PR, SILCONC\_PR

PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Compressive strength

PARACOLD(4,NM) Tensile strength ( $\geq 0$ )  
PARACOLD(5,NM) Time at which this concrete is cast.  
Before and until this time, the material does not carry  
any stress or have any stiffness.

If CMAT = CALCO\_COLD, SILCO\_COLD

PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Compressive strength,  $f_{cm}$ , in N/m<sup>2</sup>

Note: the following equations are embedded in SAFIR

If CMAT = PARABCONC

PARACOLD(1,NM) E, Young's modulus.  
PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Compressive strength  
PARACOLD(4,NM) Tensile strength  
PARACOLD(5,NM) Strain at compressive strength  
PARACOLD(6,NM) Ultimate strain

If CMAT = AL6061T6C, AL5083SUP, AL5083INF, AL7020SUP,  
AL7020INF (aluminium)

PARACOLD(2,NM)  $f_{0.2}$   
PARACOLD(3,NM)  $f_p$   
PARACOLD(4,NM)  $e_{rupture}$  in %

If CMAT = SLS1.4301, SLS1.4401, SLS1.4404, SLS1.4571,  
SLS1.4003, SLS1.4462, SLS1.4311 (stainless steel)

PARACOLD(1,NM) Young's modulus.  
PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Yield strength  
PARACOLD(4,NM) Ultimate tensile strength

If CMAT = STEELEC2EN

PARACOLD(1,NM) Young's modulus.  
PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Yield strength

PARACOLD(4,NM) Maximum temperature for a reversible behaviour during cooling.

PARACOLD(5,NM) Rate of decrease of the residual yield strength when the maximum temperature has been greater than PARACOLD(4,NM) [N/m<sup>2</sup>K]

PROCESS Chain of character that indicates the fabrication process of the reinforcing bar. It can be:

HOTROLLED for hot rolled bars (columns 2, 4 and 6 in Table 3.2a of EN 1992-1-2)

COLDWORKED for cold worked bars (columns 3, 5 and 7 in Table 3.2a of EN 1992-1-2)

CLASS Chain of character that indicates the class of ductility of the reinforcing bars. It can be:

CLASS\_A for Class A (low) ductility

CLASS\_B for Class B (high) ductility

CLASS\_C for Class C (very high) ductility

If CMAT = STEELEC3, STEELEC3EN, STEELEC3DC, STEELEC2, PSTEELA16, STEEL\_WPB

PARACOLD(1,NM) Young's modulus.

PARACOLD(2,NM) Poisson ratio.

PARACOLD(3,NM) Yield strength

PARACOLD(4,NM) Maximum temperature for a reversible behaviour during cooling.

PARACOLD(5,NM) Rate of decrease of the residual yield strength when the maximum temperature has been greater than PARACOLD(4,NM) [N/m<sup>2</sup>K]

If CMAT = STEELSL

PARACOLD(1,NM) Young's modulus.

PARACOLD(2,NM) Poisson ratio.

PARACOLD(3,NM) Yield strength

PARACOLD(4,NM) Maximum temperature for a reversible behaviour during cooling.

PARACOLD(5,NM) Rate of decrease of the residual yield strength when the maximum temperature has been greater than PARACOLD(4,NM) [N/m<sup>2</sup>K]

PARACOLD(9,NM) Slenderness of the plate where the material is present

PARACOLD(10,NM) Number of supports of the plate where the material is present  
 = 3 for oustand plates (flanges)  
 = 4 for internal plates (webs)

The STEELSL material is described in the reference: Franssen, J.M., Cowez, B., Gernay, T. (2014), "Effective stress method to be used in beam finite elements to take local instabilities into account", Fire Safety Science 11, in Proc. of the 11th IAFSS Symposium, Christchurch, New Zealand, Feb 10-14, pp. 544-557.

If CMAT = STEC3PROBA

PARACOLD(1,NM) Young's modulus.

PARACOLD(2,NM) Poisson ratio.

PARACOLD(3,NM) Yield strength

PARACOLD(4,NM) Maximum temperature for a reversible behaviour during cooling.

PARACOLD(5,NM) Rate of decrease of the residual yield strength when the maximum temperature has been greater than PARACOLD(4,NM) [N/m<sup>2</sup>K]

PARACOLD(6,NM) Value of the standard normal parameter  $\epsilon$  for the probabilistic reduction of yield strength with temperature

The STEC3PROBA material has the same expression of stress-strain relationship as steel of Eurocodes but the reduction of yield strength with temperature follows the logistic EC3-based probabilistic model proposed in: Khorasani N.E., Gardoni P., Garlock M. (2015). "Probabilistic fire analysis: material models and evaluation of steel structural members". *JSE*, 141(12).

If CMAT = WOODEC5

PARACOLD(1,NM)	Young's modulus.
PARACOLD(2,NM)	Poisson ratio.
PARACOLD(3,NM)	Compressive strength.
PARACOLD(4,NM)	Tensile strength.

If CMAT = USER\_STEEL

PARACOLD(1,NM)	Young's modulus at 20°C.
PARACOLD(2,NM)	Poisson's ratio at 20°C.
PARACOLD(3,NM)	Yield strength at 20°C.
PARACOLD(4,NM)	critical temperature (in °C) beyond which the yield strength is not fully recovered during cooling.
PARACOLD(5,NM)	the rate of decrease of the residual yield strength if the temperature has exceeded the critical temperature.

This USER\_STEEL material has the same expression of stress-strain relationship as steel of Eurocodes but it will behave at elevated temperatures according to the decreasing curves specified in the file "USER\_STEEL.TXT" that the user has to create and locate in the same folder as the input file.

In the file "USER\_STEEL.TXT",  $k_E$ ,  $k_{fy}$ ,  $k_{fp}$ ,  $e_{th}$ ,  $e_y$ ,  $e_t$  and  $e_u$  are given at different temperatures. Between two temperatures, a linear interpolation is performed by SAFIR.

$k_E$ ,  $k_{fy}$  and  $k_{fp}$  are the reduction factors at elevated temperatures relative to the values  $E$ ,  $f_y$  and  $f_p$  at 20°C.

$e_{th}$  is the value of the thermal elongation at elevated temperature.

$e_y$  is the value of the yield strain at elevated temperature.

$e_t$  is the value of the limiting strain for yield strength at elevated temperature.

$e_u$  is the value of ultimate strain at elevated temperature.

#### **Structure of the file "USER\_STEEL.TXT"**

One line.

Number\_of\_T:, NUMBER\_OF\_T

NUMBER\_OF\_T number of elevated temperatures at which the values of the reduction factors are given.

One line

T KE Kfy Kfp EPsth EPSy EPSt EPSu

One line for each temperature added to series.

T,  $k_E(T)$ ,  $k_{fy}(T)$ ,  $k_{fp}(T)$ ,  $e_{th}(T)$ ,  $e_y(T)$ ,  $e_t(T)$ ,  $e_u(T)$

T = Temperature at which the reduction factors are given

$k_E(T)$  = reduction factor relative to the value of E (Young's modulus) at 20°C

$k_{fy}(T)$  = reduction factor relative to the value of  $f_y$  (effective yield strength) at 20°C

$k_{fp}(T)$  = reduction factor relative to the value of  $f_p$  (limit of proportionality) at 20°C

$e_{th}(T)$  = thermal elongation at temperature T

$e_y(T)$  = yield strain at temperature T

$e_t(T)$  = limiting strain for yield strength at temperature T

$e_u(T)$  = ultimate strain at temperature T

**Note:**

To have the same thermal elongation as in the material STEELEC3EN for all temperatures, the first value written in the file must be equal to -1.

Example: The following file describes a material that has user defined variations of the E,  $f_y$  and  $f_p$ , but the same thermal elongation as the steel of Eurocode 3, and the same yield strain, limiting strain and ultimate strain as Eurocode 3.

Number\_of\_T: 4

T	KE	Kfy	Kfp	EPSth	EPSy	EPSt	EPSu
0.	1.0	1.00	1.00	-1.	0.02	0.15	0.20
200.	1.0	0.95	0.90	-1.	0.02	0.15	0.20
800.	0.1	0.15	0.10	-1.	0.02	0.15	0.20
1200.	0.0	0.00	0.00	-1.	0.02	0.15	0.20

## BIAXIAL PLANE STRESS MATERIAL TYPES

If CMAT = ELPLANESTR

PARACOLD(1,NM) Young's modulus.

PARACOLD(2,NM) Poisson ratio.

PARACOLD(3,NM) Coefficient of thermal expansion

Elastic plane stress material law. The material is valid for steel at elevated temperature and the Young's modulus and thermal strain vary according to the Eurocode 3 part 1.2.

If CMAT = PLSTRVML

PARACOLD(1,NM) Young's modulus.

PARACOLD(2,NM) Poisson ratio.

PARACOLD(3,NM) Yield strength

PARACOLD(3,NM) Strain hardening modulus

This model is a simplified model for steel at elevated temperature, with a bilinear equivalent stress-strain relationship. The model STEELEC32D is to be preferred, if no problem of convergence is encountered.

The parameters vary according to the Eurocode 3 part 1.2. (variation of the strain hardening modulus as for the Young's modulus).

If CMAT = STEELEC32D

PARACOLD(1,NM)	Young's modulus.
PARACOLD(2,NM)	Poisson ratio.
PARACOLD(3,NM)	Yield strength
PARACOLD(4,NM)	Maximum temperature for a reversible behaviour during cooling.
PARACOLD(5,NM)	Rate of decrease of the residual yield strength when the maximum temperature has been greater than PARACOLD(4,NM) [N/m <sup>2</sup> K]

If CMAT = STEELEC3PS

PARACOLD(1,NM)	Young's modulus.
PARACOLD(2,NM)	Poisson ratio.
PARACOLD(3,NM)	Yield strength
PARACOLD(4,NM)	Maximum temperature for a reversible behaviour during cooling.
PARACOLD(5,NM)	Rate of decrease of the residual yield strength when the maximum temperature has been greater than PARACOLD(4,NM) [N/m <sup>2</sup> K]

If CMAT = CALCOETC2D, SILCOETC2D

PARACOLD(2,NM)	Poisson ratio.
[-]	Recommended value: 0.2
PARACOLD(3,NM)	Compressive strength
[N/m <sup>2</sup> ]	
PARACOLD(4,NM)	Tensile strength ( $\geq 0$ )
[N/m <sup>2</sup> ]	
PARACOLD(5,NM)	Strain at peak stress
[-]	Recommended value: 0.0025
PARACOLD(18,NM)	Dilatancy parameter
[-]	Recommended value: 0.25 (0.20 –
0.30)	

PARACOLD(19,NM)	Compressive ductility parameter
[-]	
	Recommended value: 0.19 (0.15 – 0.25)
PARACOLD(20,NM)	Compressive damage at peak stress
[-]	
	Recommended value: 0.30 (0.18 – 0.32)
	Condition: < 0.50
PARACOLD(21,NM)	Tensile ductility parameter
[N/m <sup>2</sup> ]	
	Recommended value: 400 N/m <sup>2</sup>
	N.B. This parameter can be estimated as 100 [N.m/m <sup>2</sup> ] / where is the area of the shell element.

The CALCOETC2D, SILCOETC2D materials are plastic-damage constitutive models for concrete. The models are described in: Gernay T., Millard A., Franssen J.M. (2013). "A multiaxial constitutive model for concrete in the fire situation: Theoretical formulation". *Int J Solids Structures*, 50(22-23), 3659-3673.

If CMAT = CALCONC2D, SILCONC2D

PARACOLD(2,NM)	Poisson ratio.
PARACOLD(3,NM)	Compressive strength
PARACOLD(4,NM)	Tensile strength ( $\geq 0$ )
PARACOLD(5,NM)	
	< 0 if peak stress strain $e_{c1}$ = minimum value (stiffer)
	= 0 if peak stress strain $e_{c1}$ = recommended value
	> 0 if peak stress strain $e_{c1}$ = maximum value (more ductile)

If CMAT = LWCONC2D

PARACOLD(2,NM)	Poisson ratio
[real]	

PARACOLD(3,NM) Compressive strength  
[real]

PARACOLD(4,NM) Tensile strength  
[real]

If CMAT = VMRANK2D

PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Compressive strength  
PARACOLD(4,NM) Tensile strength ( $\geq 0$ )

If CMAT = BLPLSTRVM

PARACOLD(1,NM) Young's modulus  
PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Yield strength  
PARACOLD(4,NM) slope of the hardening branch

If CMAT = BLPLSTRDP

PARACOLD(1,NM) Young's modulus  
PARACOLD(2,NM) Poisson ratio.  
PARACOLD(3,NM) Yield strength  
PARACOLD(4,NM) slope of the hardening branch  
PARACOLD(5,NM) a

If CMAT(NM) = 'BLPLSTRVM'

PARACOLD(1,NM) = E, the Young's modulus.  
PARACOLD(2,NM) = The Poisson's ratio.  
PARACOLD(3,NM) =  $f_p$ , the limit of proportionality.  
PARACOLD(4,NM) = ???

Bi-linear plane stress Von Mises material law. The material is valid at room temperature.

## TRIAXIAL MATERIAL TYPES

If CMAT = STEELEC23D, STEELEC33D

PARACOLD(1,NM)	Young's modulus.
PARACOLD(2,NM)	Poisson ratio.
PARACOLD(3,NM)	Yield strength
PARACOLD(4,NM)	Maximum temperature for a reversible behaviour during cooling.
PARACOLD(5,NM)	Rate of decrease of the residual yield strength when the maximum temperature has been greater than PARACOLD(4,NM) [N/m <sup>2</sup> K]

If CMAT = CALCOETC3D, SILCOETC3D

PARACOLD(2,NM)	Poisson ratio. [-]
PARACOLD(3,NM)	Compressive strength [N/m <sup>2</sup> ]
PARACOLD(4,NM)	Tensile strength (≥0) [N/m <sup>2</sup> ]
PARACOLD(5,NM)	Strain at peak stress [-] Recommended value: 0.0025
PARACOLD(18,NM)	Dilatancy parameter [-] Recommended value: 0.25 (0.20 – 0.30)
PARACOLD(19,NM)	Compressive ductility parameter [-] Recommended value: 0.19 (0.15 – 0.25)
PARACOLD(20,NM)	Compressive damage at peak stress [-] Recommended value: 0.30 (0.18 – 0.32) Condition: < 0.50
PARACOLD(21,NM)	Tensile ductility parameter [N/m <sup>2</sup> ]

Recommended value: 400 N/m<sup>2</sup>

N.B. This parameter can be estimated as 100

[N.m/m<sup>2</sup>] / where is the volume of the solid element.

The CALCOETC3D, SILCOETC3D materials are plastic-damage constitutive models for concrete. The models are described in: Gernay T., Millard A., Franssen J.M. (2013). "A multiaxial constitutive model for concrete in the fire situation: Theoretical formulation". *Int J Solids Structures*, 50(22-23), 3659-3673.

SERIES 24: Time discretization.

One line, first line of possible multiple line series.

'TIME'

Time frames.

Two cases are possible:

1) In a **dynamic analysis with comeback**, a single time step must be used because the program adjusts itself the time steps during calculation.

One single line is required:

TIMESTEP, UPTIME, TIMESTEPMAX

TIMESTEP = Initial time step in seconds.

UPTIME = Time for end of the calculation.

TIMESTEPMAX = Maximum value of the time step.

2) In **other cases** several lines can be given (maximum of IDIMTIMESTEP lines, = 20 in SAFIR2007).

One line added for each time frame added.

TIMESTEP, UPTIME

TIMESTEP = Time step in seconds.

UPTIME = Limit of validity of this time step.

Time last line.

One line, end of time discretization series.

'ENDTIME'

SERIES 25: Thermal elongation.

One line, choice of two options.

'NOEPSTH'            If thermal elongation is not considered.

or

'EPSTH'            If thermal elongation is considered.

SERIES 26: Output results.

One line, first line of multiple line series.

'OUTPUT'

One line.

'TIMEPRINT'

Timeprint frames.

One line added for each timeprint frame added (maximum of  
IDIMTIMEPRINT lines)

TIMEPRINT, UPTIMEPRINT

TIMEPRINT    Time step for the output of the results.

UPTIMEPRINT    Limit of validity of this timeprint.

Timeprint last line.

One line, end of time discretization series.

'END\_TIMEPR'

Output optional results.

Add one line for each option chosen.

'PRINTDEPL'            The increments of displacement or of  
temperatures are written at every iteration.

[A9]

Tstart            Time from which the increments will be written.

[Real]

'PRINTTMPRT'            The temperatures in the fibres of the beam  
elements are written.

'PRINTVELAC'      The velocity and acceleration are written at every time step (In a dynamic analysis).

'PRINTFHE'      The out of balance forces are written at every iteration.      [A9]

Tstart      Time from which the increments will be written.  
[Real]

'PRINTREACT'      The reactions are written for at every node where at least one degree of freedom is restrained (by a BLOCK or a SAME command). The sum of the reactions of all nodes is also written for each degree of freedom. It allows verifying the total applied load (except when master-slave relationships are used for the supports, in which case the results may be confusing because some reactions are counted several times).

'PRINTMN'      Print the internal forces of the beam elements.

Axial forces are positive in tension.

Bending moments calculated as  $M_y$ , hence  $M_y$  is positive if tension prevails in the regions of the section with positive values of  $y$ .

Shear forces calculated as  $V_y$  where  $M_{y,n}$  is the bending moment at the last longitudinal point of Gauss in the beam element,  $M_{y,1}$  is the bending moment at the first longitudinal point of Gauss in the beam element, and DL is the distance between these two points of Gauss.

Similar for  $M_z$  and  $V_z$ .

'PRNSIGMASL', NSOL      Print the stresses in the solid elements.

NSOL      Number of the solid element where the mechanical strains, the stresses, the damage in tension and in compression and the accumulated plastic strain will be printed. Valid only for a 3D structural analysis.

Note: If NSOL = 0, then the stresses are printed for all solid elements. This produces a large amount of output.

'PRNEIBEAM'            Print the stiffness EA, ES and EI in the beam elements

'PRNSIGMABM', NBM, NG Print the stresses in a beam element (positive in tension).

NBM    Number of the beam element where stresses are printed.

NG     Integration point of the beam element where stresses are printed.

'PRINTET', NBM, NG            Print the tangent moduli in a beam element.

NBM    Number of beam element where moduli are printed.

NG     Integration point of the beam element where moduli are printed.

'PRNEPSMBM', NBM, NG        Print the mechanical strains in a beam element (, positive in tension).

NBM    Number of beam element where mechanical strains are printed.

NG     Integration point of the beam element where mechanical strains are printed.

'PRNSIGMASH', NSH            Print the stresses in a shell element.

NSH    Number of the solid element where the stresses are printed.

'PRINTSHELL'            Equivalent to 'PRNSIGMASH' for **all** the shell elements (large amount of results).

'PRNNXSHELL'            Print the membrane forces Nx, Ny and Nxy, N<sub>1</sub>, N<sub>2</sub> and a in the shell elements

'PRNMXSHELL'      Print the bending moments  $M_x$ ,  $M_y$  and  $M_{xy}$ ,  $M_1$ ,  $M_2$  and  $a$  in the shell elements

'PRNEASHELL'      Print the membrane stiffness  $E A_x$ ,  $E A_y$  at the 4 integration points on the surface of the shell elements (in an elastic element, this stiffness would be ).

'PRNEISHELL'      Print the bending stiffness  $E I_x$ ,  $E I_y$  at the 4 integration points on the surface of the shell elements (in an elastic element, this stiffness would be ).

'PRNSTRAIN', EPS\_LIM      Print a message (with the strain and the stress) when the absolute value of the strain in a bar of a shell element exceeds a certain limit.

                         EPS\_LIM      Limit of the strain that triggers the message.

Output results last line.

One blank line as last line of series.

### 3.4 Description and Format of the .IN file for Torsional Analysis

SERIES 1: Comments.  
One line for each comment (can be 0 line).

SERIES 2:  
One blank line to mark end of comments.

SERIES 3: Number of nodes.  
One line.  
"NNODE", NNODE  
NNODE = Number of nodes of the section.

SERIES 4: Number of axes.  
One line.  
"NDIM", NDIM  
NDIM = 2 For torsion.

SERIES 5: Does not exist anymore.

SERIES 6: Degrees of freedom.  
One line.  
"NDOFMAX", NDOFMAX  
NDOFMAX = 1 For torsion calculations.

Degrees of freedom at all the nodes.  
One line.  
"EVERY\_NODE", NDOF  
NDOF ( must be 1 for torsional calculations).

Degrees of freedom at specific nodes.  
One line for each group of nodes with specific degrees of freedom.  
"FROM", NNO1, "TO", NNO2, "STEP", NNO3, "NDOF", NDOF  
NNO1 = First node of this group of nodes.

NNO2 = Last node of this group of nodes.  
NNO3 = Node step.  
NDOF = Number of degrees of freedom for this group of nodes, 0

or 1.

**Note:**

The nodes NNO1, NNO1+NNO3, NNO1+2xNNO3,...,NNO2-2xNNO3,  
NNO2-NNO3, NNO2 have NDOF degrees of freedom

or

'REPEAT', NNO1, 'TO', NNO2, 'STEP', NNO3, 'TIME', NT  
NNO1 = First node to be repeated.  
NNO2 = Last node to be repeated.  
NNO3 = Increment.  
NT = Number of times that the nodes have to be repeated.

**Note:**

This command will create the groups:

NNO1+NNO3 , NNO1+NNO3+1 , ... NNO2+NNO3  
NNO1+2\*NNO3 , NNO1+2\*NNO3+1 , ... NNO2+2\*NNO3  
...  
...  
NNO1+NT\*NNO3, NNO1+NT\*NNO3+1, ... NNO2+NT\*NNO3

**Note:**

The active nodes where the warping function has to be calculated must have  
NDOF = 1

Two options are possible for each node where the warping function must not be  
calculated:

1. declare that the node bears 1 D.o.F., then fix it in the series 6 on  
FIXATIONS in the .STR file, see § 3.4.2
2. declare that it has 0 D.o.F., which saves times in series 6 on FIXATIONS.

End of series.

One line.  
"END\_NDOF"

SERIES 7: Torsion.

One line.  
'TORSION'

SERIES 9: Renumbering strategy.

One line, choice of options.

'NORENUM' No renumbering of the equations.

or

'RENUMPERM' Renumbering of the equations by logical permutations.

or

'RENUMGEO', NNO1 Renumbering of the equations by geometrical method.

NNO1 = Node number where geometrical renumbering will start.

NNO1 = 0 (must be typed) then renumbering started successively from all the nodes.

or

'RENUM' = RENUMGEO + RENUMPERM

or

'READRENUM' Use previous renumbering from .REN file.

SERIES 11: Number of materials.

One line.

'NMAT', NMAT

NMAT = Number of different materials.

**Note:**

If two materials have the same material law but different characteristics, it makes two different materials. e.g. C20 and C25 concrete.

SERIES 12: Number of different elements.

One line, first line of five line series.

'ELEMENTS'

Number of different elements, solid elements.

One line, second line of five line series.

'SOLID', NSOLID

NSOLID = Number of SOLID elements in the section.

Number of points for integration.

One line, third line of five line series.

'NG', NGSOLID

NGSOLID = Number of points of integration in each direction in the elements, cannot be less than 1. Greater than 3 is not recommended.

Number of voids.

One line, fourth line of five line series.

'NVOID', NVOID

NVOID = 0

Last line of series.

"END\_ELEM"

SERIES 13: The nodes.

One line, of multiple line series.

'NODES' or 'NODES\_CYL'

'NODES\_CYL' is used if cylindrical coordinates are used.

(r,q,Z) and are transformed to (X,Y,Z) for the internal solution process by the formula:

$$X = r \cos(q)$$

$$Y = r \sin(q)$$

**Note:**

q is in degrees.

The transformation is made after all the nodes have been read and generated.

CYLINDRIC is omitted if the nodes are directly input in the Cartesian system of coordinates.

NODES

One line added for each node described.

'NODE', NNO, RCOORDG(1,NNO), RCOORDG(2,NNO)

NNO = Number of the specific node.

RCOORDG(1,NNO) = First global coordinate of the node NNO.

RCOORDG(2,NNO) = Second global coordinate of the node NNO.

or 'GNODE', NNO, RCOORDG(1,NNO), RCOORDG(2,NNO)

NNO = Number of the specific node.

RCOORDG(1,NNO) = First global coordinate of the node NNO.

RCOORDG(2,NNO) = Second global coordinate of the node NNO.

This command is used for automatic equidistant generation  
between the previously defined node and node NNO

or 'REPEAT', NNO, DELTAC(1), DELTAC(2), KGENE

NNO = Number of nodes to be repeated.

DELTAC(1) = Increment on the first coordinate.

DELTAC(2) = Increment on the 2nd coordinate.

KGENE = Number of times that this command has to be repeated.

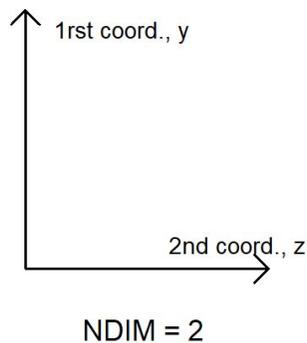


Figure 3 : Coordinate order

**Note:**

The first coordinate corresponds to the local **y** axis and the second coordinate corresponds to the local **z** axis of the beam element.

SERIES 14: Torsional centre.

One line, first line of two line series.

<A10>,<5b>,<G10.0>,<G10.0>

'NODELINE', Yo, Zo

Yo = First global coordinates of the node line which joins the  
beam elements.

Zo = Second global coordinate of the node line.

Torsional centre.

One line, second line of two line series.

'YC\_ZC', Yc, Zc

Yc = First global coordinate of the centre of torsion.  
Zc = Second global coordinate of the centre of torsion.

SERIES 15: Supports and imposed displacements.

One line, first line of possible multiple line series.

'FIXATIONS'

One line added for every node where no solution is to be calculated.

'BLOCK', NNO, 'F0'

NNO = Node number where no solution is calculated (for example, lines of symmetry).

One line added for each slave node described.

'SAME', NNO1, NNO2, CTRAV(1)

NNO1 = Number of the slave node.

NNO2 = Number of the master node.

CTRAV(1) = 'YES'

One line for repeating previous slave node.

'REPEAT', NUMBER, INC, CTRAV(1)

NUMBER = Number of times that the preceding SAME command must be repeated.

INCR = Increment on NNO1 and NNO2.

CTRAV(1) = 'YES'

Optional line to create master-slave relationships between all nodes with same coordinates

'SAMEALL', 'YES'

All the nodes of the structure that have the same coordinates (with a precision of 0.1 mm) will automatically be attributed a master-slave relationship.

Last line of series.

'END\_FIX'

SERIES 16: SOLID elements.

One line, first line of possible multiple line series.

<A10>

'NODOFSOLID'

Solid element list.

One line added for each solid element.

```
'ELEM', NE, NODE(1,NE), ..., NODE(4,NE), MAT,  
      EPSRSOLID, KGENE
```

NSOL = Number of this element.

NODE(1,NE) = First node of this element.

NODE(2,NE) = Second node of this element.

...

NODE(4,NE) = Last node of this element.

MAT = Material of this element.

EPSRSOLID = Residual stress in this element.

KGENE = Allows the generation from the previously defined element up to this one. KGENE gives the increment on the nodes number.

or

```
' REPEAT', NER, INC, NODE(2,NE), ..., NODE(4,NE), MAT,  
      EPSRSOLID, KGENE
```

NER = Number of elements to repeat.

INC = Increment on the node number.

NODE(2,NE) = Any value ( can be 0 ).

NODE(4,NE) = Any value ( can be 0 ).

MAT = Any value ( can be 0 ).

EPSRSOLID = Residual stress in this element.

KGENE = Number of times that this command has to be repeated. The element numbers increase by 1.

For triangular elements, NODE(4,NE) = 0.

**Note:**

The following group of lines on symmetry is necessary if symmetry is accounted for. If not, only the ENDSYM line is present.

Solid elements symmetry.

One line.

<A10>

```
' SYMMETRY'
```

Solid element axis of symmetry.

One line for each axis, a maximum of six axes can be specified.

<A10>,<2I5>

' REALSYM', N1, N2  
N1 = First node on axis.  
N2 = Second node on axis.

**Note:**

This line means that the line passing by the nodes N1 and N2 is an axis of symmetry. When creating the .TOR file, the fibres located on the other side of the line are created. This option is used when there is a thermal axis of symmetry, which will not be a structural axis of symmetry in the structural calculation.

Solid elements axis of symmetry, symmetric about y axis.  
One optional line.

<A10>  
' YSYM'

**Note:**

This line is used for plane beam elements, which have this symmetry. When creating the .TOR file, the area of the fibres is simply multiplied by 2.

Solid elements last line.  
One line to mark end of series and symmetry.

<A10>  
' ENDSYM'

SERIES 11: Precision.  
One line.

<A10>,<G10.0>  
'PRECISION', PRECISION  
PRECISION = Small tolerance value reached to have convergence. A 'good' value depends on the type of structure that is analyzed.  $10^{-3}$  may be used for the first simulation to look at the incremental displacements and if out of balance forces needs a different value.

SERIES 17: Material description.  
One line, first line of possible multiple line series.

<A10>  
' MATERIALS'

Material description line pair added for each different material used.

One line, first line of two line pair.

<A10>

CMAT

CMAT = Name of the material.

**Valid material names are:**

```
' ELASTIC', ' BILIN',  
' STEELEC3', 'STEELEC3EN', ' STEELEC2', 'STEELEC2EN', '  
PSTEELA16',  
'CALCONC_EN', 'SILCONC_EN', 'CALCONC_PR', 'SILCONC_PR'
```

Material description properties.

One line, second line of two line pair.

<G10.0>,<G10.0>,<G10.0>

The value of the following three parameters depends on the material name introduced in CMAT

If CMAT(NM) = ELASTIC, BILIN, or for STEEL type materials.

PARACOLD(1,NM) = Young's modulus.

PARACOLD(2,NM) = Poisson's ratio.

For the CONCRETE type materials

PARACOLD(2,NM) = Poisson's ratio.

PARACOLD(3,NM) = Compressive strength fc.

PARACOLD(4,NM) = Tension strength, not used here

The Young's modulus for concrete materials is calculated according to the formula:

SERIES 18: Output results.

One line, first line of multiple line series.

<A10>

'OUTPUT'

One line,

'TIMEPRINT'

Timeprint frames.

One line, second line of multiple line series.

TIMEPRINT, UPTIMEPRINT

<A.10>,<G.10.0>

TIMEPRINT = Any value.

UPTIMEPRINT = Any value.

Timeprint last line.

One line, end of time discretization series.

'END\_TIMEPR'

Output results last line.

One blank line to mark end of series.

<A80>

### 3.5 Structure of the .TEM files used with the BEAM F. E.

As many lines as needed

<A80>

Comment lines

1 blank line

<A80>

1 line

<A10,I5>

"NFIBERBEAM"

NFIBERBEAM(NGB) : # of fibres in this section

1 line

<A10>

" FIBERS"

1 line

<A10,2G10.0>

"NODELINE" Series for the position of the node of the beam element in the (y,z) system of co-ordinates.

system Y0

y co-ordinate of the node in the (y,z)

Z0

z co-ordinate of the node in the (y,z) system

1 line

"YC\_ZC" Series for the position of the centre of rotation of the beam element in the (y,z) system of co-ordinates.

system Yc

y co-ordinate of the centre of rotation in the (y,z)

Zc z co-ordinate of the centre of rotation in the (y,z) system

NFIBERBEAM lines

<3E13.6,I5,E13.6>

RCOORDYZINBEAM(1,NFB,NGB) : y co-ordinate of this fibre

RCOORDYZINBEAM(2,NFB,NGB) : z co-ordinate of this fibre

FIBERSECTION(NFB,NGB) : cross sectional area

of this fibre

MATBEAM(NFB) : local number of the material

present in this fibre

EPSRBEAM(NFB,NGB) : residual stress in this fibre

1 group of lines

if this group is absent, then the warping function is equal to 0 on the cross section

1 line

<A10>

" w "

NFIBERBEAM(NGB) lines

<F12.6>

WARPING(NFB,NGB) : value of the warping function in this fibre

1 line

<A8,E12.6>

" GJ= "

GJ(NGB) : torsional stiffness of the cross section

End of the group

1 line

<A10>

Either

"

COLD" : This section is not heated. The temperature in all fibres remains at 20°C. ==> The .TEM file can be ended here. No need to write the next groups ( TIME, NFIBERBEAM )

or

" HOT" : This section is heated. The next groups are necessary

1 group of lines

Repeat this group of lines for each time step

1 blank line

<A80>

1 line

<7x,F8.1>

" TIME "

1 blank line

<A80>

NFIBERBEAM(NGB) lines

<5x,F6.1>

TEMPBEAM(2,NFB,NBG) : temperature in the fibre NGB

### 3.6 Structure of the .TSH files used with the shell F. E.

#### SERIES 1 : Comments.

**Any number of lines**

```
comment cards  
<A80>
```

1 blank line indicating that the comments are finished

#### SERIES 2 : Thickness of the shells.

**1 line**

```
- "THICKNESS"  
<A9>  
- THICKSHELL      Thickness of this section type  
<Real>
```

#### SERIES 3 : Material of the shells.

**1 line**

```
- "MATERIAL"  
<A8>  
- MAT  
<Integer>
```

Local material number of this section type. This is the material of the plain section, to which layers of re-bars can be added.

#### SERIES 4 : Layers of re-bars.

**1 line**

```
- "REBARS"  
<A6>
```

- NBARS  
<Integer> # of re-bar layers in this section type.

**NBARS groups of cards.**

**1 card.**

- "MATERIAL"  
<A8>  
- MAT Local # of the material of this layer  
<Integer>

**1 card.**

- "SECTION"  
<A7>  
- A  
<Integer>  
Cross sectional area of this layer (in m<sup>2</sup>/m)

**1 card (optional)**

- "SIGMA\_RES"  
- sigma initial stress of the bars in this layer (N/m<sup>2</sup>)

**Notes:**

- 1) Residual stresses can be introduced in the bars if the bars are made of steel. If not, the stress can be entered as 0 or this card can be omitted.
- 2) Positive values are given for tension in the bars.
- 3) This model is valid for pretensioning. The stress in the bars will be reduced during the first time step because of the elastic shortenings developing in concrete.
- 4) For introducing residual stresses in the material of the plain section, see Section 3.7 'Instructions

for introducing residual stresses in steel shell elements'.

**1 card.**

- "LEVEL"  
<A5>  
- z  
<Integer>

Position of this layer with respect to the thickness

For each bar layer, there are two methods to give the orientation of the bars in the plane of the element.

**Method 1: with respect to the local system of coordinates of each element.**

**1 card.**

- "ANGLE"  
<A5>  
- angle  
<Real>

Angle in degrees between the local **x** axis and the layer of rebars, see Figure below.

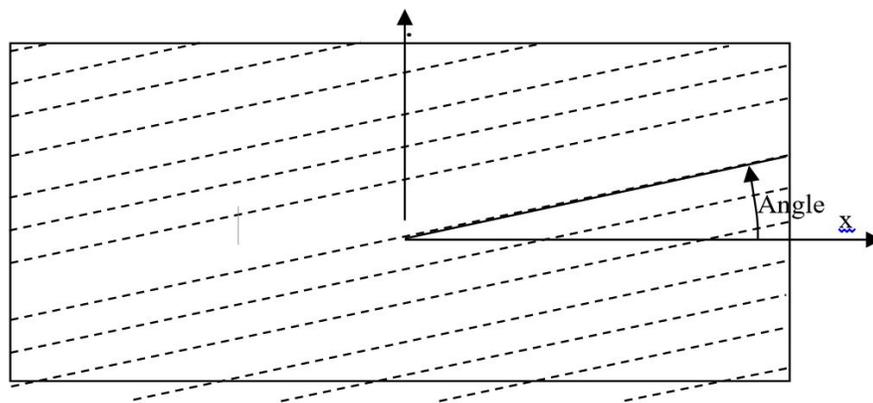


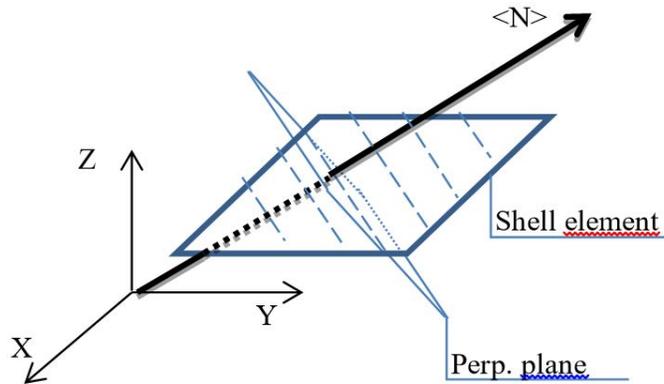
Figure 4 : Orientation of rebars in shell F.E. in local axes

**Method 2: with respect to the global system of coordinates of the structure.**

**1 card.**

- "NORMAL"  
<A6>  
-  $N_1$   
<Real>  
-  $N_2$   
<Real>  
-  $N_3$   
<Real>

$\langle N_1 ; N_2 ; N_3 \rangle$  is a vector in the global system of coordinates of the structure. The norm of the vector does not have to be 1. This vector is used to define the position of the bar layers in the shell elements with respect to the global system of coordinates according to the following technique, see figure below.



The bars have the orientation of the line which is the intersection between the shell element and a plane that is perpendicular to the normal.

If the norm of the vector is 0, then the orientation of this bar layer is perpendicular, in each element, to the previous bar layer (not possible for bar layer 1).

**SERIES 5 : Cold or Hot section**

**1 line**

Either

" COLD"

<A10>

This section is not heated. The temperature in the shell remains at 20°C. => The .TSH file can be ended here. No need to write the next series of cards

or  
" HOT" <A10>

This section is heated by a time-temperature curve (the same curve for all the elements of this type). The next groups are necessary

or  
"TuserShell" <A10>

This section is heated according to a function that has been programmed by the user in the DLL called SAFIRDLL.DLL.

No need to write the next groups

or  
" HASEMI" <A10>

This section is heated by a Hasemi fire. The next groups are necessary.

### **SERIES 6 : position of the nodes**

#### **1 line**

<A24>  
- " POSITIONS OF THE NODES."

#### **1 line**

<A25>  
- " ====="

#### **1 line**

<A21,I4>  
- " NUMBER OF POSITIONS:"  
-Number\_of\_positions

Gives the number of nodes which give the temperature of the slab across its thickness. The positions of these nodes only depends on the discretisation which was chosen when the temperature distribution was calculated. It is independent of the location of the integration points across the thickness which will be used in the structural analysis.

#### **1 line**

<Number\_of\_positions Real>  
-position of the first node (the one with the smallest z coordinate)  
-position of the second node  
- etc

- 
- position of the lest node (the one with the highest z coordinate)

**Series 7 : temperatures**

Repeat this group of lines for each time step

**1 blank line**

**1 line**

<A6, F12.4>

- "TIME="

- TIME value of the time when the temperatures are given

**1 line**

<A6>

- "====="

**Number\_of\_position lines**

<2 Real>

- position of the node (same as in series 6)
- temperature at this node

The example underneath gives the beginning of a TSH file that is 60 mm thick and has two layers of rebars of 322 mm<sup>2</sup>/m located at mid depth of the slab and perpendicular to the global Y axis (first layer) and X axis (second layer).

Safir\_Shell\_Temperature\_Analysis  
Mesh\_from\_GID-Mesher

THICKNESS 0.060  
MATERIAL 1  
REBARS 2  
MATERIAL 2  
SECTION 322e-6  
LEVEL 0  
NORMAL 0. 1. 0.  
MATERIAL 2  
SECTION 322e-6  
LEVEL 0  
NORMAL 1. 0. 0.

HOT

POSITIONS OF THE NODES.

=====

NUMBER OF POSITIONS: 6  
-0.3000E-01 -0.1800E-01 -0.6000E-02 0.6000E-02 0.1800E-01 0.3000E-01

TIME= 60.0000 SECONDS OR 1 MIN. 0 SEC.

=====

-0.0300	62.51
-0.0180	20.57
-0.0060	19.59
0.0060	20.05
0.0180	20.00
0.0300	20.00

TIME= 120.0000 SECONDS OR 2 MIN. 0 SEC.

=====

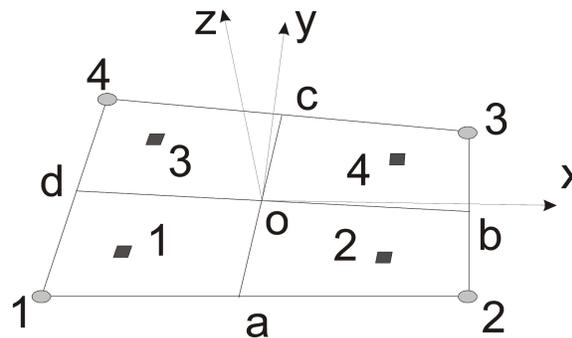
-0.0300	114.89
-0.0180	33.42
-0.0060	20.16
0.0060	19.82
0.0180	20.01
0.0300	20.01

### 3.7 Instructions for introducing residual stresses in steel shell elements

This section explains how to consider residual stresses in the first material of the shell element. This procedure has been foreseen to introduce residual stresses in steel plates, not in the rebars of reinforced concrete slabs. For the latter, see SERIES 4 in Section 3.6 'Structure of the TSH files'.

In order to introduce residual stresses, it is necessary to create a file with the name "*resi\_str\_shell.txt*", which must be placed in the same folder as the structural input file.

In this file, the residual stresses should be placed in Pascal, for each point of integration of the elements in which there are residual stresses.



- Nodes
- Points of integration

**Figure 5: Position of the integration point in the plane**

There are 4 points of integration in the plane of the element located as indicated by Figure 1. The integration in the plane is by the method of Gauss.

The number of integration points on the thickness NGTHICK is chosen by the user, from 2 to 9. The integration on the thickness is also by the method of Gauss.

Format of the file "*resi\_str\_shell.txt*":

One line for shell element.  
'ELEM', NE,

NE = Number of this element.

One line for each gauss point of the shell element. There are (4 x NGTHICK) Gauss points.

'PG', NPG, Sx, Sy, Sxy

NE = Number of this gauss point.

Sx = Residual strength in x axis.

Sy = Residual strength in y axis.

Sxy = Residual tangential strength.

or for automatic generation of the gauss points

'Gpg', NLPG, KGENE

NLPG = Number of the last gauss point with automatic generation.

KGENE = Step for the automatic generation of the gauss points.

or for automatic generation of the elements

'GELEM', NLE, KGENE

NLE = Number of the last element with automatic generation.

KGENE = Step for the automatic generation of the elements.

**Example:**

```
ELEM 1
  PG 1   100000   0   0
  PG 2   100000   0   0
  PG 3   100000   0   0
  PG 4   100000   0   0
  PG 5   100000   0   0
  PG 6   100000   0   0
  PG 7   100000   0   0
  PG 8   100000   0   0
ELEM 2
  PG 1  -100000   0   0
  PG 2  -100000   0   0
  PG 3  -100000   0   0
  PG 4  -100000   0   0
  PG 5  -100000   0   0
  PG 6  -100000   0   0
  PG 7  -100000   0   0
  PG 8  -100000   0   0
ELEM 3
```

PG 1	-100000	0	0
PG 2	-100000	0	0
PG 3	-100000	0	0
PG 4	-100000	0	0
PG 5	-100000	0	0
PG 6	-100000	0	0
PG 7	-100000	0	0
PG 8	-100000	0	0
ELEM 4			
PG 1	100000	0	0
PG 2	100000	0	0
PG 3	100000	0	0
PG 4	100000	0	0
PG 5	100000	0	0
PG 6	100000	0	0
PG 7	100000	0	0
PG 8	100000	0	0

Or with automatic generation:

ELEM 1			
PG 1	100000	0	0
Gpg 8	1		
GELEM 4	3		
ELEM 2			
PG 1	-100000	0	0
Gpg 8	1		
GELEM 3	1		

### 3.8 Structure of the temperature files used with the truss F. E.

As many lines as necessary, each line being, in a free format, a pair of values in the form:

TIME            TEMPERATURE

Example:

	0.	20.
	300.	600.
600.	800.	
1200.	1000.	
1500.	900.	
1800.	20.	
3600.	20.	