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

USER'S MANUAL FOR SAFIR 2011

A COMPUTER PROGRAM FOR ANALYSIS OF STRUCTURES SUBJECTED TO FIRE

by

J.M. Franssen[1]

January 2011

[1] jm.franssen@ulg.ac.be

TABLE OF CONTENTS

List of Figures

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)[1,2].

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

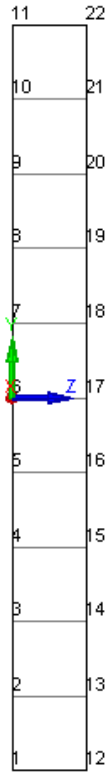
2-D SOLID elements, to be used later on cross sections of BEAM elements or on the thickness of SHELL elements.

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.



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
  MATERIAL    1
    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 three different element types: Truss elements, made of one single material with one uniform temperature per element; beam elements, either pure steel, reinforced concrete or composite-steel sections; and shell 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. This means that there is no deflection criterion to actually make the failure point.

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` $f = 2\,t / 60$
- `F20` $f = 20$
- `F100` $f = 100$
- `F1000` $f = 1000$
- `F1000PS` $f = 1000\,t$
- `F1PSM1000` $f = 0$ for $t \le 1000$; $f = t - 1000$ for $t > 1000$
- `F1000THPS` $f = t / 1000$
- `FLOAD` $f = t / 20$ for $t \le 20$; $f = 1$ for $t > 20$
- `FISO` $f = 20 + 345 \log_{10}(8\,t / 60 + 1)$
- `FISO0` $f = 345 \log_{10}(8\,t / 60 + 1)$
- `HYDROCARB` $f = 20 + 1080\,(1 - 0.325\,e^{-0.167\,t/60} - 0.675\,e^{-2.5\,t/60})$

- `ASTME119` *f* defined by linear interpolation between a set of (time ; temperature) pairs, with time in minutes.

Time	Temp.	Time	Temp.	Time	Temp.	Time	Temp.	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℃ 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 $y_i \sigma_i$, with y_i measured on the local axis L1, are positive when fibres having a positive local coordinate are in tension.

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 SafirWizard (for special cases only), 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 **.TEM** files created during thermal and torsional analyses and in which the temperature data is stored.

Figure 1 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 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.

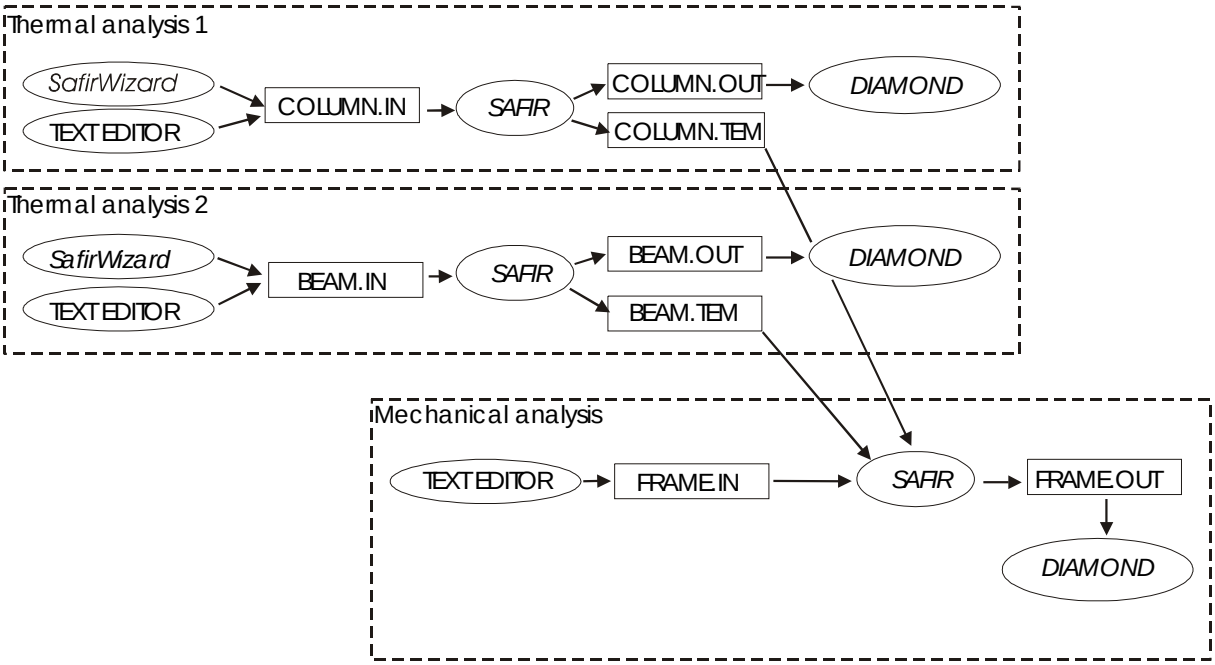


Figure 1 : files and steps

2.3 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"		
"NDOFMAX"		
"EVERY_NODE"		Optional
"FROM" **OR** "REAPEAT"	FROM N1 TO N2 STEP N3 NDOF N4 REPEAT N1 TO N2 STEP N3 TIMES N4	Optional, multiple lines possible
"END_NDOF"	end of the section	
"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
"NORENUM" **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"		Multiple lines possible. Last I5 not present if NODE command is used

“FIXATIONS”		
“BLOCK”		Optional, multiple lines possible
“SAME”, "SAMEALL" **OR** “REPEAT”		Optional, multiple lines possible
"END_FIX"	line for end of section	
“NODOFBEAM”		For beam elements
Filename.TEM	A20	Left justified file name of **.TEM** 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
Filename.TSL		File name for **.TSL** file related to solid elements
“ELEM” or "GELEM" **OR** “REPEAT”		Multiple lines possible for solid elements
“ ENDSYM”		For solid elements
“NODOFSHELL”		Optional
Filename.TSH	<A20>	File name for **.TSH** 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
Filename.TRS	<A20><3*G10.0><I5>	File name for **.TRS** file related to truss elements
	<6*I5>	Nodes of truss elements
“ 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
“PRINTSOLID”		Optional
“PRINTVELAC”		Optional
“PRNSIGMASH”		Optional
“PRNNXSHELL”		Optional
“PRNEASHELL”		Optional
“PRNEISHELL”		Optional
“PRNSIGMABM”	<I><I>	Optional
“ PRINTET”	<I><I>	Optional
" PRNEPSMBM	<I><I>	Optional
"PRNEIBEAM"		optional
	Blank line for end of comments	

2.4 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 and X_GYPSUM (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, USER_STEEL, CALCONCEC2, SILCONCEC2, LWCONCEC2, SILCONC_EN, CALCONC_EN, SILCON_ETC, CALCON_ETC, 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, SILCONC2D, CALCONC2D, ELPLANESTR, PLSTRVML, BLPLSTRVM, BLPLSTRDP, VMRANK2D (these materials have 2D plane stress 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 (ε_i), thermal effects (ε_{th}) and stress related effects (ε_σ). The stresses are, therefore, caused by the difference between the total strain (ε_{total}), obtained from the nodal displacements, and the initial and thermal strains.

2.5 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

Notes:

1. This series has the effect of allocating NDOF degrees of freedom (D.o.F.) to all the nodes. If all the nodes of the structure have the same number of D.o.F., then the "END_NDOF" line finishes the series. Otherwise additional lines have to be used to include the nodes which have a different number of D.o.F. The subsequent lines do this. If the line with the command 'EVERY_NODE' is not present, then all the nodes have 0 D.o.F. and the following lines have to be used to attach D.o.F. to the nodes where they must be present.
2. The active nodes where the temperature is calculated must have NDOF = 1.
3. Two possible options for nodes where temperatures are not calculated:

- declare that a node has 1 D.o.F., then fix it in series 6 on FIXATIONS in the .STR file.
- declare the a node has 0 D.o.F. This saves time in series 6 on FIXATIONS.

Note:

If NDIM = 2 **and** the results of a thermal analysis are to be used in the structural analysis of beam elements, then the first coordinate corresponds to the local **y** axis of the beam element and the second coordinate corresponds to the local **z** axis of the beam element.

The residual stress is meaningful only in the case of a thermal analysis made on the plane cross section of a beam, the results of which will be used in a structural analysis. In other cases, any value can be specified, preferably 0.

Notes:

A fire curve can be a predefined function (e.g., “`FISO`”), or a file name.

The heat flux at the boundary will be calculated from the temperature of the fire curve T_g and the temperature on the surface T_s according to $q = h\left(T_g - T_S\right) + \sigma \varepsilon^* \left(T_g^4 - T_S^4\right)$

Note:

"F20" frontiers can be added on other surfaces, e.g. on the upper unheated side of a slab.

If the user does not know the value to start with, 10^{-3} is used to make a first simulation and look at the incremental displacements and out of balance forces. This can give some useful information to lead to a precision estimate. See series 13.

Note : This value is also used as a small number at different locations in SAFIR. A good value depends on the type of structure that is being analyzed and the experience of the user. If the user does not know the value to start with, 10^{-3} is used to make a first simulation and look at the incremental displacements and out of balance forces. This can give some useful information to lead to a precision estimate. See series 13.

Note: Thermal conductivity, specific heat, and specific mass are taken from EN for STEELEC3EN and STEELEC2EN, from ENV for the other steels. The only difference is that, for ENV steels, the emissivity is divided by 0.8 in internal cavities (VOIDS)

Note: For ENV concretes, the emissivity is divided by 0.8 in internal cavities (VOIDS)

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 6: 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 ≥ 2
for solid elements, NDOFMAX ≥ 2
for beam elements, NDOFMAX ≥ 3
if NDIM = 3 for truss elements, NDOFMAX ≥ 3
for solid elements, NDOFMAX ≥ 3
for shell elements, NDOFMAX ≥ 6
for beam elements, NDOFMAX ≥ 7

Degrees of freedom for all the nodes.
"EVERY_NODE", NDOF
NDOF = Number of degrees of freedom

Note :
- This line allocates NDOF D.o.F. to all the nodes.
- If all the nodes of the structure have the same number of D.o.F., then the "END_NDOF" line mentioned below finishes the series. If not, additional lines have to be used to mention the nodes which have a different number of D.o.F. The following lines do this.
- If the line with the command 'EVERY-NODE' is not present, then all the nodes have 0 D.o.F. and the following lines have to be used to attach D.o.F. to the nodes where they must be present.

Degrees of freedom for a series of nodes.
One line added for each series of nodes.
"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 group of nodes.

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

or

One line added for a repeating series of nodes, repeated copies get the same degrees of freedom as the original.
'REPEAT', NNO1, ' TO', NNO2, ' STEP', NNO3, ' TIME', NT
NNO1 = First node to be repeated.
NNO2 = Last node to be repeated.
NNO3 = Node step.
NT = Number of times that the nodes are to be repeated.

Note:
The command will create the following groups:

NNO1+NNO3,	NNO1+NNO3+1,	...	NNO2+NNO3
NNO1+2xNNO3,	NNO1+2xNNO3+1,	...	NNO2+2NNO3
.....		...	.....

.....	...	.....
NNO1+NTxNNO3,	NNO1+NTxNNO3+1, ...	NNO2+NTxNNO3

If NDIM = 2:
The nodes supporting truss and solid elements must have NDOF ≥ 2:
- translation in the global axis 1
- translation in the global axis 2

The end nodes supporting beam elements must have NDOF ≥ 3:
- translation in the global axis 1
- translation in the global axis 2
- rotation about virtual global axis

The internal node of beam elements must have NDOF = 1:
- 2nd order component of the longitudinal displacement, no other elements must be linked to this node

If NDIM = 3:
The nodes supporting truss and solid elements must have NDOF ≥ 3:
- translation in the global axis 1
- translation in the global axis 2
- translation in the global axis 3

The end nodes supporting beam elements must have NDOF ≥ 7:
- translation in the global axis 1
- translation in the global axis 2
- translation in the global axis 3.
- rotation about global axis 1.
- rotation about global axis 2.
- rotation about global axis 3.
- warping.

The internal node of beam elements must have NDOF = 1:
- 2nd order component of the longitudinal displacement

The nodes of shell elements must have NDOF = 6:
- translation in the global axis 1
- translation in the global axis 2
- translation in the global axis 3
- rotation about global axis 1
- rotation about global axis 2
- rotation about global axis 3

One line.

"END_NDOF"
Indicates that the series is finished

SERIES 5 (optional): MATRIX SOLVER

1 card (optional)

SOLVER, CHOLESKY

- SOLVER [A6]

  command

- 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*

- NCORES [A6]

  command

- *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 recommanded for structures made of beams, and "APPR_NR" is recommanded 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 the load that will vary with time according to the same function.

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.

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

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 $Kq + C\dot{q} + M\ddot{q} = F$ 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 then 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 then three iterations, the time step is multiplied by 2 (limited at the value of the initial time step).

SERIES 11 (optional): EQUATION RENUMBERING

One (and only one) of the following cards may be used. The card may be omitted if the sparse matrix solver has been used (which is the default option), because renumbering is perform by Pardiso and the renumbering made by SAFIR does not accelerate the runs; the time spent for renumbering is thus wasted. One of these cards should be used only of CHOLESKY has been chosen as the matrix solver, see series 5.

1 card (optional)

- NORENUM [A7]

  Command. No renumbering of the equations will be performed. Introducing this card has the same effect as omitting this card. The possibility is given to introduce this card for compatibility with earlier version of the GID-SAFIR interface.

1 card (optional)

- RENUMPERM [A9]

  Command. Renumbering of equations by logical permutations.

1 card (optional)

RENUMGEO, *nno1*

- RENUMGEO [A9]

  Command. Renumbering of equations by geometrical method.

- *nno1*

  [integer]

  Number of the node where geometrical renumbering will start. If *nno1* = 0, then renumbering starts successively from all nodes (this process can take a long time).

  RENUMGEO, *nnoi* with *nnoi* being the number of a node located geometrically in a corner of the model is probably a good compromise between the time spent for renumbering and the acceleration provided to the run in the method of CHOLESKY.

1 card (optional)

- RENUM [A4]

  Command. Has the same effect as RENUMGEO, *1* + RENUMPERM. Produces then most efficient renumbering, but can take a very long time.

1 card (optional)

- READRENUM [A8]

  Command. The renumbering has been done in a previous run. The match between equation numbers before renumbering and equation numbers after renumbering is read from the .REN file that had been created during the renumbering. This card can be used only if the typology of the model has not been modified since the renumbering has been made.

SERIES 12: 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 13: Number of different elements.
One line, first line of multiple line series.
'ELEMENTS'

Different elements, beam elements sub-series.
One line added to sub-series if beams are used in the structure.
'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.

Different elements, integration points, beam elements sub-series.
One line.
'NG', NG
NG = Number of longitudinal points of integration in elements. Cannot be less than 2. Greater than 3 is not recommended.

Different elements, fibres of beam elements, beam elements sub-series.
One line.
'NFIBER', NFIBERBEAM
NFIBERBEAM = Number of longitudinal fibres in the beam elements (the maximum value for all the different groups of geometrical properties).

Different elements, truss elements.
One line added to series if truss elements are used.
'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.

Different elements, shell elements sub-series.
One line, first line of four line sub-series.
'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.

Different elements, Shell elements sub-series shell thickness.

One line, second line of four line sub-series.
'NGTHICK', NGSHELLTHICK
NGSHELLTHICK = Number of points of integration on the thickness of the elements. Cannot be less than 2 and cannot be more than 9.

Different elements, shell element sub-series rebar.

One line, fourth line of four line sub-series.
'NREBARS', NREBARS
NREBARS = Number of REBAR layers in the shell elements.

Last line of series.

"END_ELEM"

SERIES 14: 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
$(r,\theta) \Rightarrow X = r\cos(\theta), Y = r\sin(\theta)$. if NDIM = 2
$(r,\theta,Z) \Rightarrow X = r\cos(\theta), Y = r\sin(\theta), Z,$ if NDIM = 3

Note:
θ 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.
…
…
DELTAC(NDIM) = Increment on the coordinate NDIM.
KGENE = Number of times that this command has to be repeated.

SERIES 15: 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,NN0) 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.

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.

Last line, indicating that the series is finished

‘END_FIX’

SERIES 16: 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) = 4th 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    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 17: SOLID elements. Not possible at the time being because no 3D material model has been implented and validated.

SERIES 18: 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>
'NODOFSHELL'

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 19: 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 20-0: Oblique supports

One line for each oblique support.
`'inclin', Ni, Nj` if NDIM = 2 (2D structure)
`'inclin', Ni, Nj, Nk` if NDIM = 3 (3D structure)
`Ni` is the node where a boundary condition is expressed in a local system of coordinates.
`Nj, Nk` are 2 other nodes of the structure.
Ni and Nj, for a 2 D structure, or Ni, Nj and Nk, for a 3D structure, define the plane in which the node Ni can move. It cannot move perpendicularly to this plane.

One line to indicate that this is the end of the Series 10-0 <A10>
`'END_INCLIN'`

SERIES 21: Precision.

One line.

'PRECISION', PRECISION

PRECISION = Small value that must be reached for convergence.

Note:

This value is also used as a small number at different locations in SAFIR. A good value depends on the type of structure that is being analyzed and the experience of the user. If the user has no idea of the value to start with, try 10^{-3} ($5*10^{-4}$ in case of dynamic analysis) then make a first simulation and look at the displacement and Out of Balance Forces (see SERIES 18). This can give some useful information that could lead to a good precision estimate.

SERIES 22: Loading.

One line, first line of possible multiple line series.

'LOADS'

Loading function.

One line, second line of possible multiple line series.

'FUNCTION', CFORCE(NLO)

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

Loading function possible multiple node loads.

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

Loading on beam elements.

One line added for each element with a 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

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, last line of series.
'END_LOAD'

SERIES 23: 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 kgm².
2) Usually, a concentrated mass linked to a displacement is active in all directions. Only in soma 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)

Mass on beam elements (Self-Weight).

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 of the beam element section $= \sum_i \rho_i \, I_{p,i}$

$$\text{With} \begin{cases} I_{p,i} = I_{y,i} + I_{z,i} + y_{CG,i}^2 \, A_i + Z_{CG,i}^2 \, A_i \\ \rho_i = \text{Volumic mass of the material i} \end{cases}$$

(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 of the beam element section

(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)

Mass on shell elements (Self-Weight).

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/m²).

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/m²).

KGENE Increment on the element number

Mass end of series.

One, last line of series.

'END_MASS'

SERIES 24: 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
- ELASTIC, CALCONC_EN, SILCONC_EN, CALCON_ETC, SILCON_ETC, CALCONC_PR, SILCONC_PR, CALCONCEC2, SILCONCEC2, LWCONCEC2, PARABCONC, CALCO_COLD, SILCO_COLD, STEELEC3, STEELEC3EN, STEELEC3DC, 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
- ELPLANESTR, PLSTRVML, STEELEC32D, SILCONC2D, CALCONC2D, VMRANK2D, BLPLSTRVM, BLPLSTRDP

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, 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℃.)

PARACOLD(1,NM) E, Young's modulus.
PARACOLD(1,NM) Poisson ratio
PARACOLD(3,NM) lp, the limit of proportionality.
PARACOLD(4,NM) n, exponent of the law.
PARACOLD(5,NM) K, factor of the law.

$$\varepsilon = \frac{\sigma}{\mathrm{E}} \qquad \text{for } \sigma \le l\mathrm{p}$$

$$\varepsilon = \frac{\sigma}{\mathrm{E}} + \left(\frac{\sigma - l\mathrm{p}}{\mathrm{K}} \right)^{\mathrm{n}} \qquad \text{for } \sigma > l\mathrm{p}$$

If CMAT = CALCONCEC2, SILCONCEC2

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

If CMAT = CALCONC_EN, SILCONC_EN, LWCONCEC2

PARACOLD(2,NM) Poisson ratio.
PARACOLD(3,NM) Compressive strength
PARACOLD(4,NM) Tensile strength

If CMAT = CALCON_ETC, SILCON_ETC

PARACOLD(2,NM) Poisson ratio.
PARACOLD(3,NM) Compressive strength
PARACOLD(4,NM) Tensile strength

If CMAT = CALCONC_PR, SILCONC_PR

PARACOLD(2,NM) Poisson ratio.
PARACOLD(3,NM) Compressive strength
PARACOLD(4,NM) Tensile strength
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/m²
Note: the following equations are embedded in SAFIR

$$f_t = 0,0$$

$$E = 1,05 \times 22\,000 \left(\frac{f_{cm}}{10} \right)^{0,3} \quad \text{for} \quad \text{SILCO\_COLD}$$

$$E = 1,05 \times 0,9 \times 22\,000 \left(\frac{f_{cm}}{10} \right)^{0,3} \quad \text{for} \quad \text{CALCO\_COLD}$$

$$\varepsilon_{c1} = 0,7\, f_{cm}^{0,31} \times 10^{-3} \leq 2,8 \times 10^{-3}$$

$$\varepsilon_{cu} = 3,5 \times 10^{-3} \quad \text{if} \quad f_{cm} \leq 50$$

$$\varepsilon_{cu} = \left(2,8 + 27 \left(\frac{98 - f_{cm}}{100} \right)^4 \right) \times 10^{-3} \quad \text{if} \quad f_{cm} > 50$$

with f_{cm} in MPa

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) $\varepsilon_{rupture}$ 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 = STEELEC3, STEELEC3EN, STEELEC3DC, STEELEC2, STEELEC2EN, 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/m²K]

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} and ε_{th} 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.
ε_{th} is the value of the thermal elongation 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`

One line for each temperature added to series.

T, $k_E(T)$, $k_{fy}(T)$, $k_{fp}(T)$, $\varepsilon_{th}(T)$
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
$kf_{y(}T)$ = reduction factor relative to the value of fy (effective yield strength) at 20°C
$k_{fp}(T)$ = reduction factor relative to the value of fp (limit of proportionality) at 20°C
ε_{th} (T) = thermal elongation 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.

```
Number_of_T:  4
T     KE     Kfy     Kfp     EPSth
   0.  1.    1.      1.      -1.
 200.  1.    0.95    0.90    -1.
 800.  0.1   0.15    0.1     -1.
1200.  0.    0.      0.      -1.
```

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/m²K]

If CMAT = `CALCONC2D, SILCONC2D`

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

If CMAT = `VMRANK2D`

PARACOLD(2,NM) Poisson ratio.
PARACOLD(3,NM) Compressive strength
PARACOLD(4,NM) Tensile strength

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) α

If CMAT(NM) = ‘BLPLSTRVM’

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

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

If CMAT(NM) = ‘STEELEC3PS’

PARACOLD(1,NM) = E, the Young’s modulus.
PARACOLD(2,NM) = The Poisson’s ratio.
PARACOLD(3,NM) = fy, the limit of proportionality.

Non-linear plane stress material law. The material is the steel material according to Eurocode 3 part 1.2.

SERIES 25: 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 27: Thermal elongation.
One line, choice of two options.

'NOEPSTH' If thermal elongation is not considered.

or

'EPSTH' If thermal elongation is considered.

SERIES 28: 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 displacement variation is written at every iteration.

'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.

'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 $M_y = \int_\Omega \sigma\, y\, d\Omega$, hence M_y is positive if tension prevails in the regions of the section with positive values of *y*.

Shear forces calculated as $V_y = \frac{M_{y,n} - M_{y,1}}{\Delta L}$ 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 ΔL is the distance between these two points of Gauss.
Similar for M_z and V_z.

'PRINTSOLID' Print the stresses in the solid elements.

'PRNSIGMABM', NBM, NG Print the stresses in a beam element.
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 ($\varepsilon_m = \varepsilon_{tot} - \varepsilon_{th}$).
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_1, N_2 and α in the shell elements

'PRNMXSHELL' Print the bending moments Mx, My and Mxy, M_1, M_2 and α in the shell elements

'PRNEASHELL' Print the membrane stiffness EAx, EAy at the 4 integration points on the surface of the shell elements (in an elastic element, this stiffness would be $\frac{E\,t}{1-\nu^2}$).

'PRNEISHELL' Print the bending stiffness EIx, EIy at the 4 integration points on the surface of the shell elements (in an elastic element, this stiffness would be $\frac{E\,t^3}{12\left(1-\nu^2\right)}$).

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

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,θ,Z) and are transformed to (X,Y,Z) for the internal solution process by the formula:
$X = r \cos(\theta)$
$Y = r \sin(\theta)$

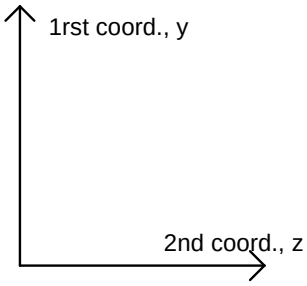
Note:
θ is in degrees.
The transformation is made after all the nodes have been read and generated.
CYLINDRIC is omitted if the nodes are directly input in the Cartesian system of coordinates.

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.



NDIM = 2

Figure 2 : 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:

$$E = \frac{2f_c}{2.5\times10^{-3}}$$

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.
Y0 y co-ordinate of the node in the (y,z) system
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.
Yc y co-ordinate of the centre of rotation in the (y,z) system
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"
- THICKSHELL Thickness of this section type

SERIES 3 : Material of the shells.
1 line
- "MATERIAL"
- MAT 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"
- NBARS # of re-bar layers in this section type.

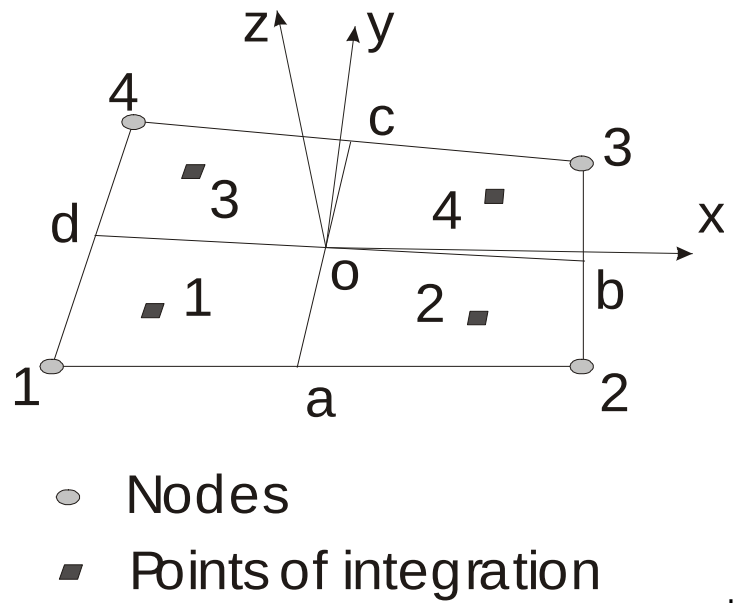
NBARS groups of cards.

1 card.
- "MATERIAL"'
- MAT local # of the material of this layer

1 card.
- "SECTION"
- A cross sectional area of this layer (in m^2/m)

1 card.
- "LEVEL"
- z position of this layer with respect to the thickness

1 card.
- "ANGLE"
- angle Angle in degrees between the local ***x*** axis and the layer of rebars, see Figure below



SERIES 5 : Cold or Hot section

1 line <A10>

Either

" COLD" : 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 groups

or

" HOT" : 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" : This section is heated according to the function that has been programmed by the user in the DLL called SAFIRDLL.DLL. No need to write the next groups

or

" HASEMI" : 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_position : 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

- position of the first node (the one with the smallest z co-ordinate)
- position of the second node
- etc
-
- position of the lest node (the one with the highest z co-ordinate)

Series 7 : temperatures

Repeat this group of lines for each time step

1 blank line

1 line

- "TIME="
- TIME value of the time when the temperatures are given

1 line

"=====“

Number_of_position lines

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

3.7 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.