2 INPUT

FAHTS (Fire And Heat Transfer Simulations) reads input from symbolic files.

The user may give all input on one file or distribute the data on two files. All control parameters are specified in the Analysis Control File. Structure data can also be read from this file, but is usually given on one separate file. The specific content of these files is not important, as long as all data are present.

These files may be written with a text editor or generated by pre-processors.
FAHTS reads the data required by the temperature analysis only, and data not needed by FAHTS are just disregarded.

FAHTS reads the USFOS structure files and exports the temperature data (BELTEMP) to be used by USFOS. In addition to the USFOS structure model file format, FAHTS also reads the SESAM FEM-files generated by GeniE.

The input records specific for FAHTS temperature analysis are presented in Section 2.3. Structure input is presented in Section 2.4, (see also USFOS User's Manual for more complete description).

2.1 GENERAL INFORMATION

The input data are organized in records, each record starting with a record identificator of four to eight characters. Each record may consist of one or more lines of data, terminating on the next record identificator. Each line may be **up to 132 characters long**. The data items may be integer on real data.

The data records may be given in an arbitrary order.

In this manual, each record is presented in a standard frame. Each frame represents either one single record or a sequence of similar records.

2.2 FORMATS

Nearly all input data are read by FREAD /6/, a FORTRAN free-format reader and decoder. This means that the data items may be written anywhere on the line, **as long as the specified order is satisfied.** The data items must be separated with at least one blank (exceeding blanks are ignored). Note that blank is exclusively interpreted as a delimiter and cannot be used to specify a zero value as accepted by standard FORTRAN READ.

Important:

All digits, letters and/or special symbols in a data item must be given consecutively without blanks.

2.2.1 Comments

Lines with an apostrophe (') in the first column are interpreted as comments, and simply ignored. Comments may occur anywhere in the input data stream.

Example:

```
'THIS IS A COMMENT
' NOTE!  COMMENTS ARE IGNORED BY THE PROGRAM
```

2.2.2 Alphanumeric Data Items

An alphanumeric data item may consist of one or more characters. The first character is always a letter (A-Z), while the remaining ones may be letters, digits or special symbols (except /, $, & and blank).

There is no upper limit to the number of characters in an alphanumeric data item. However, only the first 8 characters will be decoded, and all characters in excess of this are simply ignored.

2.2.3 Integer Data Items

- All characters must be digits
- The first digit may be preceded by + or -

Example:

0 1 -27 +66

2.2.4 Real Number Data Items

A real-number data entry may consist of up to 3 components, i.e. an integer part i, a decimal part d, and exponent part e. The following 4 basic forms are accepted:

(±)i (±)i. (±)i.d (±).d

These may all be combined with exponent parts yielding the forms:

(±)iE(±)e (±)i.E(±)e (±)i.dE(±)e (±).dE(±)e

Example:

0 +1. -0.2E14 +17.E-3 1.8E+3

2.2.5 Text Strings

Text strings may consist of one or more characters, which may be letters (A-z), digits or special symbols.

72 characters are stored in a text string, beginning at the 9th character of the line.

Example:

This text string uses special characters & \ #

And will be stored as

t string uses special characters & \ #

2.2.6 Arithmetic Operations

The FII input reader interprets some simple numerical operations. This means that the user may define mathematical expressions in the input, for example in order to scale / add parameters. The expressions are:

- Adding (+)
- Subtraction (-)
- Multiplication (*)
- Division (/)
- Trig (SIN(ang) and COS(ang), ang in radians)

Example. Adding value to coordinate:

```
`     ID     X       Y         Z
NODE  10    0.0   10.0+1.23   10.0   ! Add 1.23 to the Y-coordinate
```

2.3 FAHTS INPUT RECORDS

This section describes the input records specified for FAHTS transient temperature analysis. Only records marked *) are mandatory, the rest are optional.

The following records are specified on the Analysis Control File:

Description	Command		Page
Analysis Identification	**HEAD**	*)	2-8
Analysis Control	**TEMPSIM**	*)	2-8
	CRESTART		2-8
Initial Conditions	**INITEMP**		2-9
	NODETEMP		2-9
Thermal Data	**THERMPAR**	*)	2-10
	INSIDPAR		2-11
	INSULPAR		2-11
Thermal Data vs. Elements	**ELMTHERM**		2-12
	ELMINSID		2-12
	ELMINSUL		2-13
	INS_GRUP		2-13
Thermal "Loads"	**KAMELEON**		2-14
	USERFIRE		2-15
	USERFLUX		2-16
	USERJET		2-17
	USERFLARE		2-18
	USERTEMP		2-19
	ISOFIRE		2-21
	HCFIRE		2-22
Thermal "Load" Option	**TIMEHIST**		2-23
	TMPCASE1		2-24
	MOVFIRE		2-24
Temperature Mesh Refinement	**TEMPMESH**		2-25
	MESHPIPE		2-26
	MESHBOX		2-27
	MESHIPRO		2-28
	MESHQUAD		2-29
Shape factor Calculations	**SHAPFACT**		2-30

Limit FEM–MODEL for Temperature Calculations	**LIMITMOD**	2-31
Limit Elements exposed to fire	**LIMTFIRE**	2-34
Temperature Dependency		
	TEMPDEPY	2-37
	THERMDEP	2-38
	INSULDEP	2-38
No Linearization of Temperature Field	**NOLINEAR**	2-39
Linearization of Temperature Field	**LIN_RULE**	2-40
Temperature plot of specified points	**TEMP_PLO**	2-41
PARTLY Exposed / Insulated Members		
Exposure of elements (all sections)	**EXP_ELEM**	2-42
Exposure of I-Sections	**EXP_IPRO**	2-43
Exposure of BOX sections	**EXP_BOX**	2-44
Exposure / PFP on quad elements	**EXP_QUAD**	2-45
Exposure of Groups	**EXP_GRUP**	2-46
Begin- and End time for using KFX data	**KAMELEXP**	2-47
Unexposed elements	**NOEXPOSURE**	2-48
Uneven U-Values over the cross section	**PFPSCALE**	2-49
No Exposure of limited cross section area	**EXPLOCAL**	2-50
Openings in PFP / PFP on limited area	**PFPLOCAL**	2-51
MISCELLANEOUS		
Visualization of groups in xact	**FatGroup**	2-52
Preparation for push down analysis in USFOS	**PushDown**	2-52
Re-defining analysis options	**SWITCHES**	2-53
Defining radiation shield	**SHIELD**	2-54
Automatic re-meshing of I-profiles	**ReMesh_I**	2-55
Automatic re-meshing of Box-profiles	**ReMesh_B**	2-55

The input records are presented in a standard frame with the following format:

RECORD IDENTIFICATOR	**Parameter No. 1**	**Parameter No. 2**	**Parameter No. 3**	**...**
Parameter	*Description*			
1	Description of contents and default values			
2	Description of contents and default values			
3	Description of contents and default values			
Optional box for comments, notes, exceptions etc.				

Analysis Identification

HEAD	<		>
	<	Three lines of text identifying the analysis	>
	<		>

Character 9 to 80 from each line are stored as text strings.

This record is given only once.

Analysis Control

TEMPSIM end_time nstep res_inc

Parameter	*Description*
end_time	Termination time, seconds
nstep	Total number of time steps to be used. (The time increment used during the time integration is end_time/nstep)
res_inc	Time between result save, (seconds). *BELTEMP* data and RAF file data are written to file at the specified time interval.

With this record, the user controls the temperature simulations and result storage.

This record is given only once.

CRESTART

Parameter	*Description*

With this record, FAHTS will read the pre-processed FEM-model from the specified file, (file with the extension .dms) and perform the temperature simulations only.

Following data may be updated at restart run

- TEMPSIM
- USERFIRE
- USERJET

This record is given only once.

Initial Conditions

INITEMP T0 (GradY GradZ ListType IDs …)

Parameter	*Description*	*Default*
T0	Initial mean temperature of the structure. (Default: 20 degrees Celsius.)	20
GradY	Initial temperature gradients	0
GradZ		0
ListType	Definition on how to define the elements	
IDs	List of IDs	

With this record, the user specifies the initial temperature of the structural elements.

If this record is omitted, the default temperature is used.

Note! Temperature data is given in Celsius degrees always!

This record may be repeated.

Example:

```
'               T0    GradY   GradZ    ListType   Id
 IniTemp      1200      0       0       Elem       1
 IniTemp        30      0       0       Elem       2
```

In the FAHTS analysis, the element starts with 1200° C and element 30 with 30°C. All other elements start with temperature 20°C.

NODETEMP temp inode1 inode2 ....

Parameter	*Description*
temp	Initial temperature of the nodes
inode1, ...	Actual nodes, (External node numbers used in the FEM-model)

With this record, the user specifies the initial temperature at the specified FEM-model nodal points.

Note! Temperature data is given in Celsius degrees always!

This record may be repeated.

Thermal Data

THERMPAR therm-no rho c k emiss

Parameter	*Description*
therm-no	User defined (external) number. This number has to correspond with the material number referred to in the record GELREF1
rho	Mass density, (kg/m3)
c	Specific heat capacity, (J/kgK)
k	Heat conductivity, (W/mK)
emiss	Emissivity coefficient

With this record, the user specifies the thermal properties of the materials referred to in the structural FEM-model, (at the "GELREF1" record).

Note! Data should be given according to SI-units if possible. If other units are used, all input must be given according to the actual unit. The Stefan-Boltzmann's constant used in FAHTS assumes SI-unit. The emissivity coefficient must be multiplied with the square of the ratio between actual length unit and 1 meter if other units are used!

All data in the KFX database are stored in SI-units, and if the KFX option is used in FAHTS, all input to FAHTS, (structural data and thermal data) must be given according to SI-units!

This record may be repeated.

INSIDPAR insid-no rho c

Parameter	*Description*
insid-no	User defined (external) number of this "inside-member" property parameters
rho	Mass density, (kg/m3)
c	Specific heat capacity, (J/kgK)

With this record, the user specifies the thermal properties of the material inside hollow members, (Pipe and box members).

Note! Data should be given according to SI-units if possible. If other units are used, all input must be given according to the actual unit.

This record may be repeated.

INSULPAR insul-no type K emiss PFP_{crack_ang}

Parameter	*Description*
insul-no	User defined (external) number of this insulation definition
type	Insulation type (Eq. to 1)
K	Resultant heat conduction (W/m2 K)
emiss	Emissivity factor of the surface of the insulation
PFP_{crack_ang}	The PFP is valid for a beam rotation up to this angle [deg]

With this record, the user specifies the thermal properties of the insulation used to protect structural members.

Note! Data should be given according to SI-units if possible. If other units are used, all input must be given according to the actual unit.

The Stefan-Boltzmann's constant used in FAHTS assumes SI-units. The emissivity coefficient must be multiplied with the square of the ratio between actual length unit and 1 meter if other units are used!

All data in the KFX database are stored in SI-units, and if the KFX option is used in FAHTS, all input to FAHTS (structural data and thermal data) must be given according to SI-units!

This record may be repeated.

Thermal Data vs. Elements

ELMTHERM therm-no ielex1 ielex2 ielex3 ...	
Parameter	*Description*
therm-no	User defined (external) number of the thermal data which should be connected to the specified elements
ielex1	User defined (external) element number
ielex2	User defined (external) element number
With this record, it is possible to let elements with same material reference in the structural FEM-model have different thermal properties. This record is used to override the material reference specified in the GELREF1 record. This record may be repeated.	

ELMINSID insid-no ielex1 ielex2 ielex3 ...	
Parameter	*Description*
insid-no	User defined (external) number of the inside data which should be connected to the specified elements
ielex1	User defined (external) element number
ielex2	User defined (external) element number
With this record, the user specifies the elements which should have the actual "inside-member" properties. This record may be repeated.	

ELMINSUL insul-no ielex1 ielex2 ielex3 ...

Parameter	*Description*
insul-no	User defined (external) number of the insulation data which should be connected to the specified elements
ielex1	User defined (external) element number
ielex2	User defined (external) element number

With this record, the user specifies the elements, which should use the actual insulation. All surfaces of the specified elements will be "covered" with the actual insulation by default. (Local cross section PFP could be defined using the "EXP_.." and/or "PFP_.." commands.)

If no elements are specified, all elements are connected to the actual insulation number. Additional *ELMINSUL* records with specification of elements with different insulation are possible.
This record may be repeated.

INS_GRUP insul-no Group1 Group2 ...

Parameter	*Description*
insul-no	User defined (external) number of the insulation data which should be connected to the specified element-groups
Group1	Group ID 1
Group2	Group ID 2

With this record, the user specifies the element groups, which should use the actual insulation. All surfaces of the specified elements will be "covered" with the actual insulation. (Local cross section PFP could be defined using the "EXP_.." and/or "PFP_.." commands).

This record may be repeated.

Thermal "Loads"

KAMELEON (accur)	
Parameter accur	*Description* Ray-tracing accuracy parameter. If omitted: accur = 2 is used. The accuracy parameter (optional) defines the number of "beams" used from each point when the radiation flux is calculated. Typical values: 2 - 5, (increased accuracy value results in higher accuracy) (Number of "beams" per point is: (3*accur) * (4*accur)).

With this record the user specifies that the environmental temperatures and radiation are read from the KFX result database.

FAHTS asks the user for the KFX database **prefix** when this record is specified.

The KFX results have to be stored on files using following name convention:

```
prefix_001.k2f        : The first KFX file to use in the FAHTS simulation
prefix_002.k2f        : The 2nd (if transient fire).
```

For example if the prefix is Case01:

```
Case01_001.k2f
Case01_002.k2f
Case01_003.k2f
```

FAHTS will change files automatically for Time = ($T_i + T_{i+1}$) / 2 , where T_i is the time stored on k2f file no "***i***', and $+T_{i+1}$ is the time on the next k2f file.

NOTE! All data in the KFX database are stored in SI-units, and if the KFX option is used in FAHTS, all input to FAHTS, (structural data and thermal data) must be given according to SI-units!

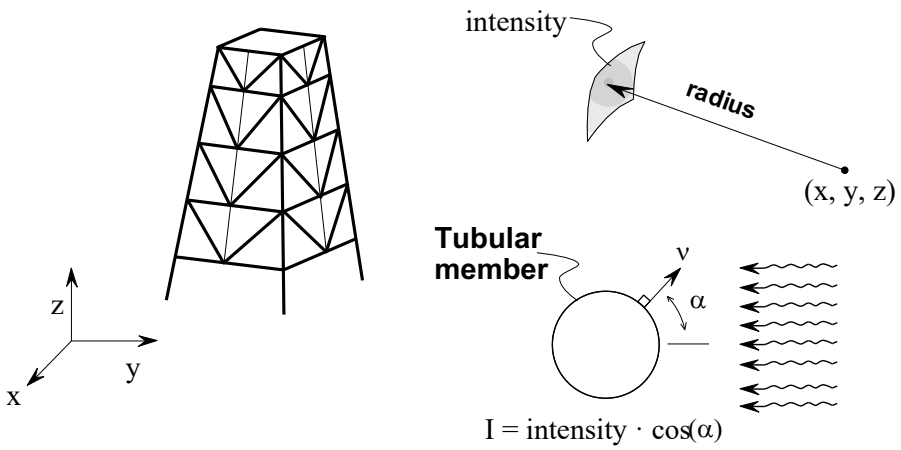
This record is given once

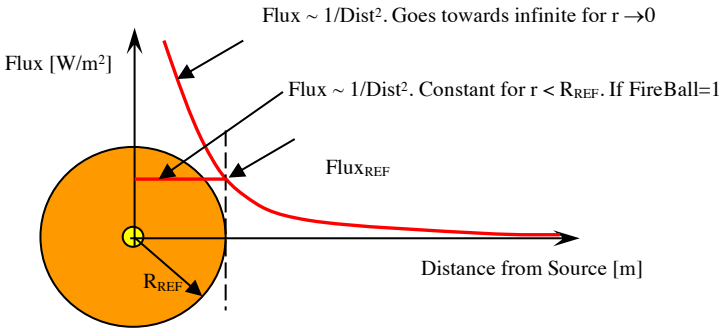
USERFIRE TimeHist x y z intensity radius (FireBall)

Parameter	*Description*
TimeHist	Time history number used to scale the intensity
x, y, z	x. y and z coordinates of the radiating source. Coordinates are given in the GLOBAL system, (system used to describe the structural geometry)
intensity	Intensity, (W/m2) at "radius" distance from the source
radius	Reference radius from the source used to define the intensity
(FireBall)	If "FireBall" is set to 1, the flux will be set to "Intensity" for all parts of the structure with distance to the source less than "Radius". Inside the "Ball", 100% engulfed is assumed (no "cosine" effect).

With this record, the user specifies a radiating source used for simplified fire scenarios which may be constant (timehist = 0) or vary as a function of time according to the specified time history.

This record could be repeated, (fluxes are added).



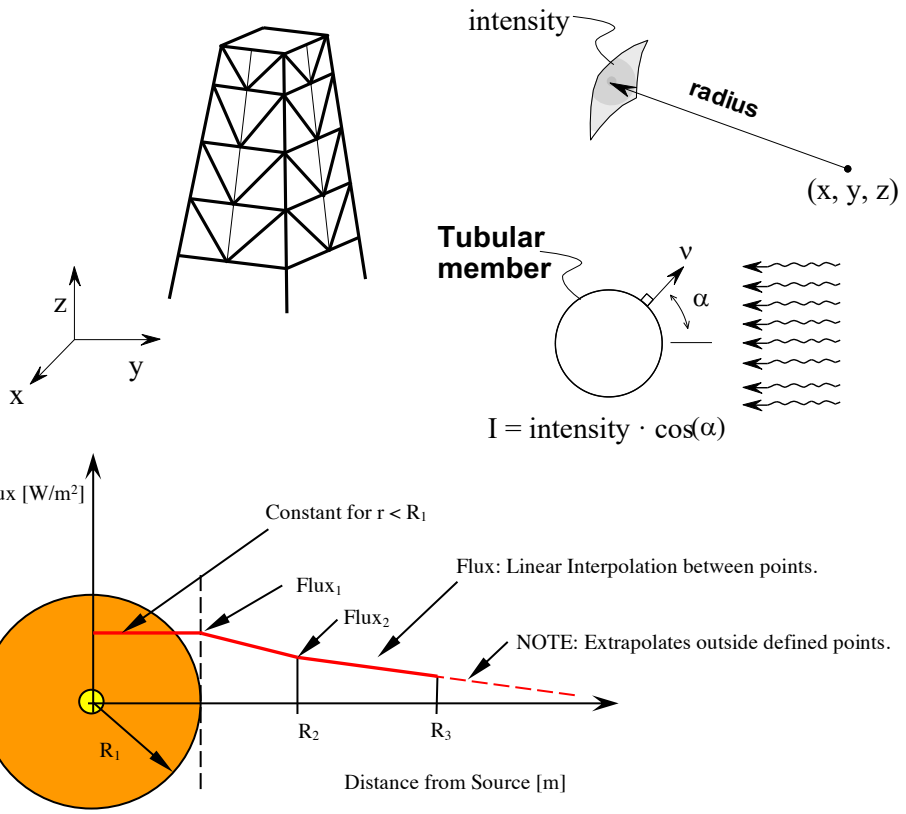


USERFLUX TimeHist Type x y z R1 Flux1 R2 Flux2 .....

Parameter	*Description*
TimeHist	Time history number used to scale the intensity. 0 means no time scaling.
Type	Flux Field Type. Type = 1: Flux defined by discrete points
x, y, z	x. y and z coordinates of the radiating source. Coordinates are given in the GLOBAL system, (system used to describe the structural geometry)
R1	Radius from the source to first point.
Flux1	Intensity, (W/m2) at "R1" distance from the source. This flux is also used points closer to source. (Assuming engulfed structure)
R2	Radius from the source to 2nd point.
Flux2	Intensity, (W/m2) at "R2" distance from the source
Ri, Fluxi	Radius/Flux to point i. NOTE: Max 7 points.

With this record, the user specifies a Flux Field defined by discrete points (Radius , Flux), used for simplified fire scenarios. Could be constant (TimeHist = 0) or vary as a function of time according to the specified time history.

This record could be repeated, (fluxes are added).

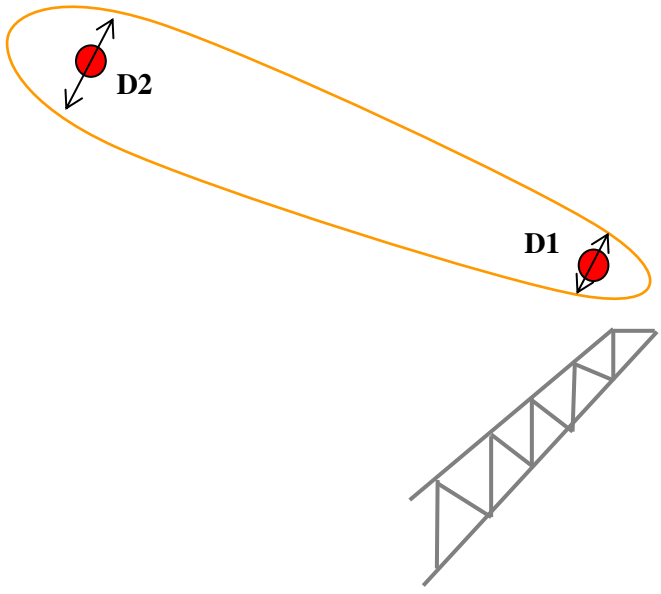


USERJET TimeHist x1 y1 z1 pow1 x2 y2 z2 pow2 (D1 D2)

Parameter	*Description*
TimeHist	Time history number used to scale the intensity
x1, y1, z1	x, y and z coordinates of end 1 of the 'conical' source. Coordinates are given in the GLOBAL system, (system used to describe the structural geometry)
Pow1	Power per length unit, [W/m] at end 1 of the 'jet'-source.
X2, y2, z2	x, y and z coordinates of end 2 of the 'conical' source. Coordinates are given in the GLOBAL system, (system used to describe the structural geometry)
Pow2	Power per length unit, [W/m] at end 2 of the 'jet'-source.
D1 D2	"Diameter" of the line-source at end 1 and end 2. The flux inside the 'flame' is set equal to the flux at the surface of the line source. If omitted, the diameter is set by FAHTS to L/10, L is length of line-source.

With this record, the user specifies a jet-source used for simplified fire scenarios which may be constant (TimeHist = 0) or vary as a function of time according to the specified time history.

This record is given once.

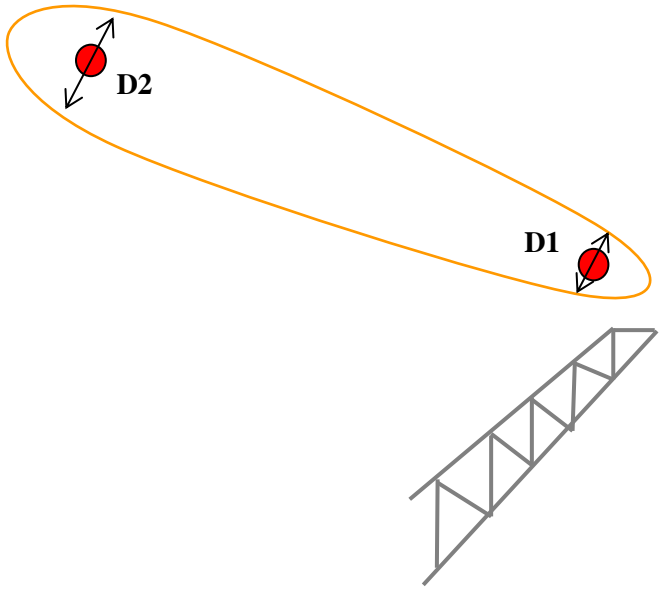


USERFLARE TimeHist Rate Energy fact x1 y1 z1 x2 y2 z2 D1 D2

Parameter	*Description*
TimeHist	Time history number used to scale the intensity
Rate	Release rate [kg/s]
Energy	Heat energy value per kg [J/kg]
fact	Fraction of the released heat emitted as radiation.
x1, y1, z1	x, y and z coordinates of end 1 of the 'conical' source. Coordinates are given in the GLOBAL system, (system used to describe the structural geometry)
x2, y2, z2	x, y and z coordinates of end 2 of the 'conical' source..
D1 D2	"Diameter" of the line-source at end 1 and end 2. The flux inside the 'flame' is set equal to the flux at the surface of the line source. If omitted, the diameter is set by FAHTS to L/10, L is length of line-source

With this record, the user specifies a jet-source used for simplified fire scenarios, which may be constant (TimeHist = 0) or vary as a function of time according to the specified time history. This "fire" is in principle the same as USERJET. The difference is that FAHTS computes the power in End-1 and End-2 of the jet.

This record is given once.



USERTEMP TimeHist temp (*1 LimitType { LimitMod Data }***)**

Parameter	*Description*
TimeHist	Time history number used to scale the temperature
temp	Environmental temperature level. Default temperature levels is 20 degrees Celsius
(Opt Data)	Data used to limit the env. temp. If no opt data are given, all elements are using the specified env-temperature.

With this record, the user specifies the environmental temperature level, which may be constant, (TimeHist = 0) or vary as a function of time (according to the specified time history).

Note! Temperature data is given in ***Celsius*** degrees always!

This record may be repeated. Later definitions override previous definitions.

If the Temperature field should be limited to certain volume, the "LIMITMOD" data could be given and the temperature is used for elements inside the actual volume.

Following "Limit" options are available:

LimitType = 1 : Box defined by two points in space (see LimitMod)
LimitType = 2 : Plane defined by 3 points and the distance (------ " -------)
LimitType = 3 : List of elements (------ " -------)
LimitType = 6 : Cone defined by two endpoints (x,y,z) and the two cone radius.
LimitType = 7 : Sphere defined by centre (x,y,z) and diameter

Examples:

```
'          Hist  Temp  GasAbs LimitType  x1   y1  z1   x2   y2  z2
 USERTEMP   1     400     1      Box   -100 -100  0  100  100  100 !  General Field
 USERTEMP   1     800     1      Box    -40  -40  0   40   40   60 !  800 Deg box
 USERTEMP   1    1000     1      Box   - 20  -20  0   20   20   40 ! 1000 Deg box
```

All elements within the first "box" are given the environmental temperature of 400 deg. Then all elements within box 2 are given environmental temp=800 deg. Finally, the elements inside the smallest box get an environmental temp of 1000 deg. Elements (midpoint coordinates are used),

Similar using three concentric spheres:

```
'          Hist  Temp  GasAbs LimitType  x1   y1  z1  Diam
 USERTEMP   1     400     1   Sphere     10   20  5    20  !  General Field
 USERTEMP   1     800     1   Sphere     10   20  5    10  !  800 Deg sphere. Diam=10m
 USERTEMP   1    1000     1   Sphere     10   20  5     7  ! 1000 Deg sphere. Diam= 7m
```

USERTEMP TimeHist Temp *1* *TempBall Typ nSph* *(Hist X Y. Z Diam)*$_1$
(Hist X Y. Z Diam)$_2$
.....
(Hist X Y. Z Diam)$_n$

Parameter	*Description*
TimeHist	Time history number used to scale the temperature. Set to zero
Temp	Dummy temperature. Set to 1
1	Parameter set to 1
TempBall	Defines that a series of concentric spheres defines the temperature field.
Typ	TempBall fire type. Is set to 6.
nSph	Number of concentric spheres to be defined thereafter
Hist	Time history controlling the diameter of the sphere. 0 means constant
X Y Z	Coordinates to the centre of the sphere
Diam	Diameter of the sphere
	Repeat for the nSph spheres.

With this record, the user specifies the environmental temperature level within spheres. The sphere diameter could be constant, (TimeHist = 0) or vary as a function of time (according to the specified time history).

Example:

```
'        Hist Temp gs  LimitType Typ  nSph Hist Temp X   Y   Z  Diam
 USERTEMP  0    1   1   TempBall  6    4     0   400 5  7.7  5   50  ! Ambient
                                           50   700 5  7.7  5    1  !
                                          100   880 5  7.7  5    1  !
                                          200  1100 5  7.7  5    1  !
'
'          ID            Time  Diam
 TimeHist  50   Points     0   14        ! Initial diameter = 14m
                           2   10        ! Shrinks to 10m after 2 minutes
                           5    5
                          10    1.0
                          25    0.3      ! Shrinks to 0.3m after 25min and is kept
                          30    0.3      ! constant thereafter

--- similar input for TimeHist 100 and 200 ----
```

All elements within the first "sphere" are given the environmental temperature of 400°C. This sphere has a diameter of 50m and is kept constant. Temperatures are given in ***Celsius*** degrees.

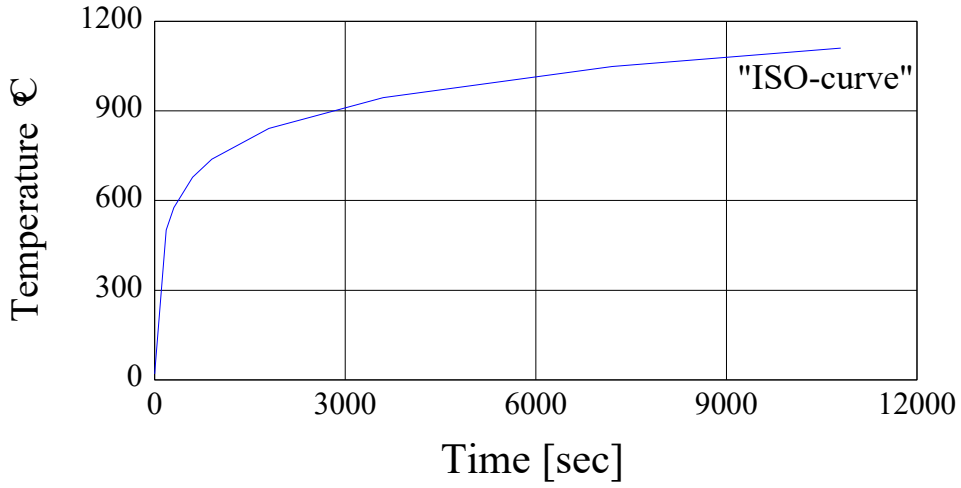
Temperature Elements within the smaller spheres get temperatures 700, 880 or 1,100°C depending on their coordinates. The *temperature* element's midpoint coordinates are used.

The Diameters of the three last spheres are changed according to histories 50, 100 and 200. The diameters are set to 1.0 for these spheres since the actual diameter vs. time are specified under TimeHist.

ISOFIRE

Parameter	*Description*
	With these record the user specified that the environmental temperature field follows the standard ISO temperature curve, as specified in the ISO-834, see figure below

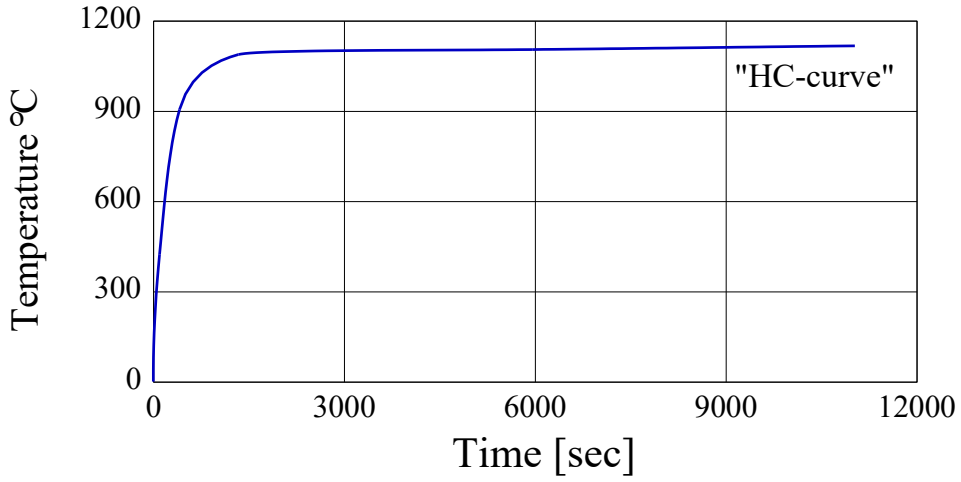
This record is given once.



HCFIRE

Parameter	*Description*
	With these record the user specifies that the environmental temperature field follows the standard Hydro Carbon fire temperature development defined by NPD, see figure below

This record is given once.



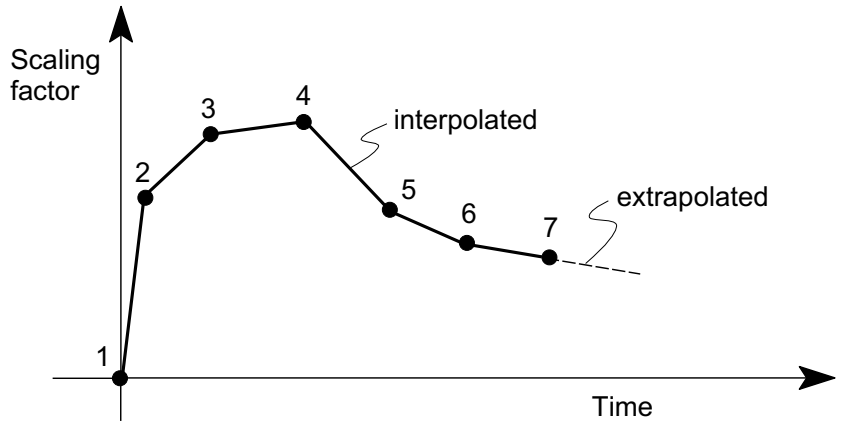
Thermal "Load" Options

TIMEHIST **histno type time1 factor1 time2 factor3 ...**	
Parameter	*Description*
histno	Time history number (user defined ID)
type	Time Hist Type = 1, (according to USFOS definition)
time1 factor1	time (seconds) Scale factor 1
time2 factor2	time (seconds) Scale factor 2

With this record, the user specifies a time history by discrete points (maximum 25 points).

Values between the tabulated points are interpolated; values outside the specified range are extrapolated as indicated in the figure below.

This record may be repeated.



TMPCASE1 lcase

Parameter	*Description*
lcase	Load case number of the first *BELTEMP* load case number generated by FAHTS. Succeeding load cases are increased by one. Default: lcase = 4

This record is given once.

MOVFIRE dX dY dZ

Parameter	*Description*
dX, dY, dZ	The KFX fire is 'moved' dX, dY and dZ relative to the structure.

With this record, the user may 'move' a KFX to the specified location.
(For example when the coordinate systems for structure and KFX have different location of origin.)

This record is given once.

Temperature Mesh Refinement

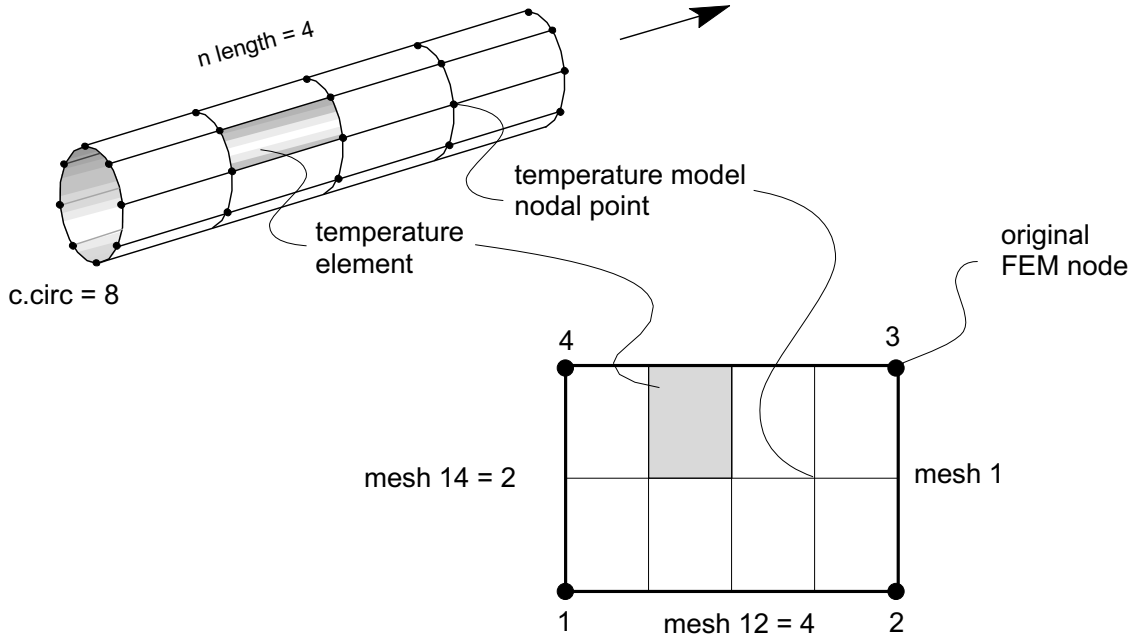
TEMPMESH nLength nCirc mesh_12 mesh_14

Parameter	*Description*
nLength	Number of 4-node quadrilateral elements in the length direction of beam elements. Default = 2
nCirc	Number of 4-node quadrilateral elements in the circumferential direction of tube/beam elements. Default = 8
mesh_12	Number of 4-node quadrilateral elements in the 1-2 direction of membrane elements. Default = 1
mesh_14	Number of 4-noded quadrilateral elements in the 1-4 direction of membrane elements. Default = 1

With this record, the user specifies the temperature mesh refinement. The mesh is automatically generated by FAHTS as a pre-processing task.

This record is override by the MESH.... records.

This record is given only once.



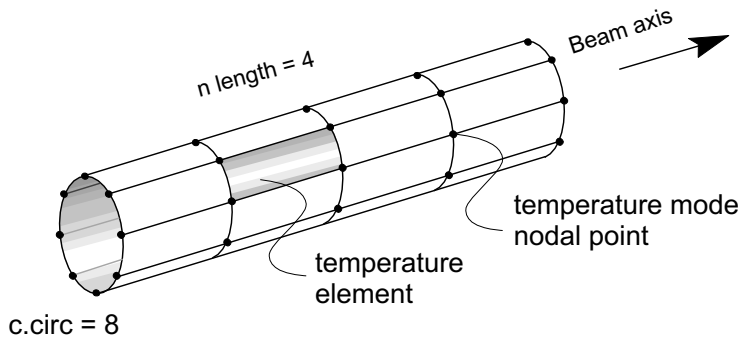
Membrane element

MESHPIPE nLength nCirc iel1 iel2 iel3 .....

Parameter	*Description*
nLength	Number of 4-node quadrilateral elements in the length direction of beam elements. Default = 2
nCirc	Number of 4-node quadrilateral elements in the circumferential direction of tube/beam elements. Default = 8
iel1 ...	Element(s) to be meshed as defined. If no elements are specified, all pipes are meshed as specified.

With this record, the user specifies the temperature mesh refinement for pipe cross-sections. The mesh is automatically generated by FAHTS as a pre-processing task.

This record may be repeated.

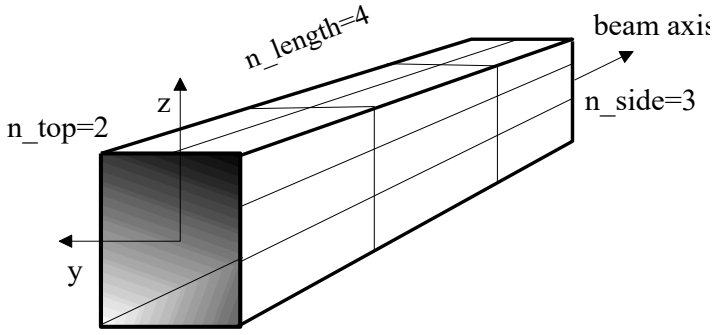


MESHBOX nLength nSide nTop nBott iel1 iel2 iel3 .....

Parameter	*Description*
nLength	Number of 4-node quadrilateral elements in the length direction of beam elements. Default = 2
nSide	Number of elements in the vertical direction of box elements. Default = 1
nTop	Number of elements in the horizontal direction of the box top 'flange'. Default = 1.
nBott	Number of elements in the horizontal direction of the box bottom 'flange'. Def = 1.
iel1 ...	Element(s) to be meshed as defined. If no elements are specified, all box profiles are meshed as specified.

With this record, the user specifies the temperature mesh refinement for box cross-sections. The mesh is automatically generated by FAHTS as a pre-processing task.

This record may be repeated.

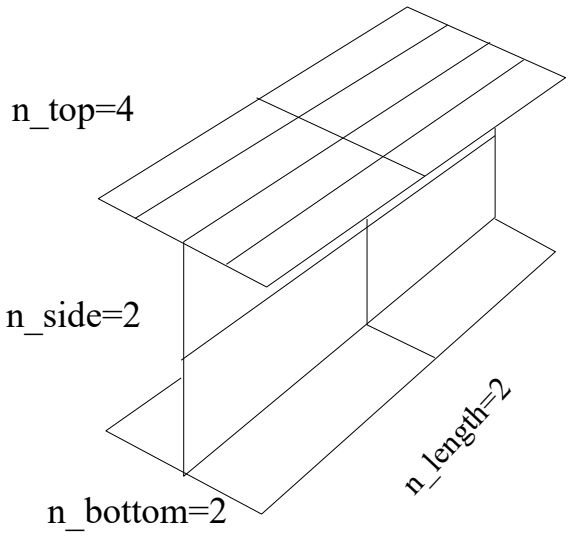


MESHIPRO nLength nSide nTop nBott iel1 iel2 iel3 .....

Parameter	*Description*
nLength	Number of 4-node quadrilateral elements in the length direction of beam elements. Default = 2
nSide	Number of elements in the vertical direction of the web. Default = 1
nTop	Number of elements in the horizontal direction of the top flange. Default = 2.
nBott	Number of elements in the horizontal direction of the bottom flange. Def. = 2.
iel1 ...	Element(s) to be meshed as defined. If no elements are specified, all I-profiles are meshed as specified.

With this record, the user specifies the temperature mesh refinement for I-cross-sections. The mesh is automatically generated by FAHTS as a pre-processing task.

This record may be repeated.

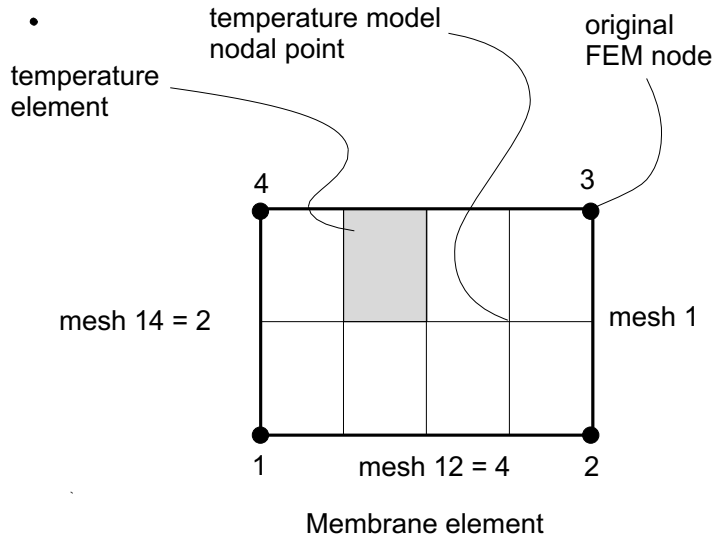


MESHQUAD n_12 n_14 iel1 iel2 iel3 .....

Parameter	*Description*
n_12	Number of elements in direction 1-2 of membrane elements. Default = 1
n_14	Number of elements in direction 1-4 of membrane elements. Default = 1
iel1 iel2 ...	Element(s) to be meshed as defined. If no elements are specified, all membrane elements are mesh as specified.

With this record, the user specifies the temperature mesh refinement for membrane elements. The mesh is automatically generated by FAHTS as a pre-processing task.

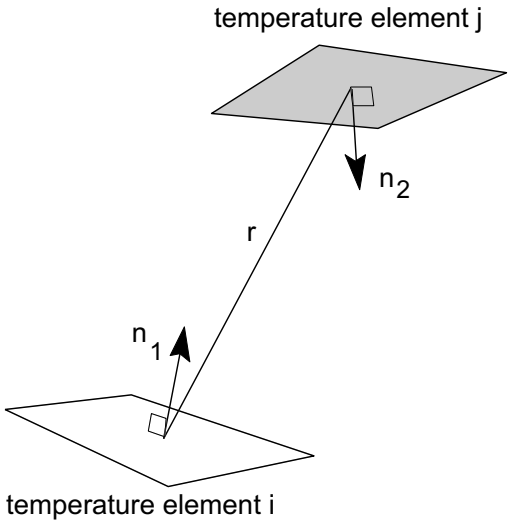
This record may be repeated.



Membrane element

Shape factor Calculations

SHAPFACT (*Tol*)	
Parameter	*Description*
	With this record, the automatic shape factor calculations is switched ON which means that radiation between temperature elements belonging to the same FEM structural element are included (radiation inside tubes ...). Shape factors (view factors) of each temperature element is calculated automatically based on a numerical integration approach.
Tol	If Tolerance is specified, the shape factors are checked against all other finite elements. If the local shape factor between two elements is greater than ***Tol***, the actual element will be included in the radiation exchange. Default: OFF



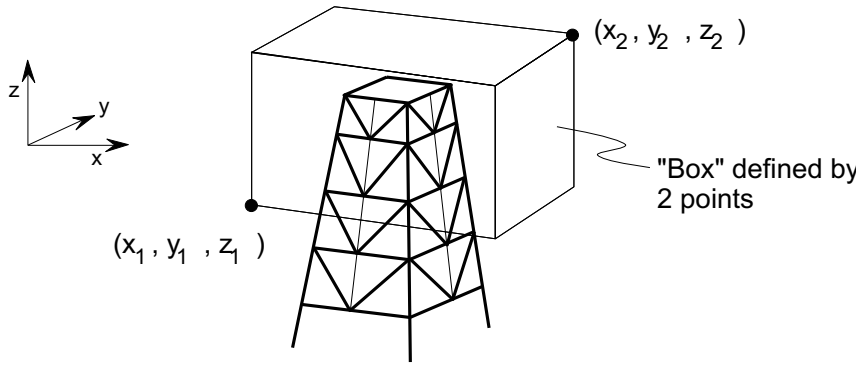
Limit FEM–model for Temperature Calculations

LIMITMOD type x1 y1 z1 x2 y2 z2

Parameter	*Description*
type	Description type: Type = 1 : Elements inside box are included Type = -1 : Elements outside box are included
x1, y1, z1	Coordinates of point 1 used to define the limitation "box"
x2, y2, z2	Coordinates of point 2 used to define the limitation "box" See figure below. Default: Entire structure included.

With this record, the user may extract the part of the entire structure, which is of interest with respect on temperature distribution. All elements inside the defined "box" (type = 1) or outside the "box" (type = -1) will be included in the transient heat transfer analysis, and *BELTEMP* records will be generated for each of these elements. Elements, which do not fulfil the limitation-test are disregarded. (and are not visualized).

This record may be repeated.

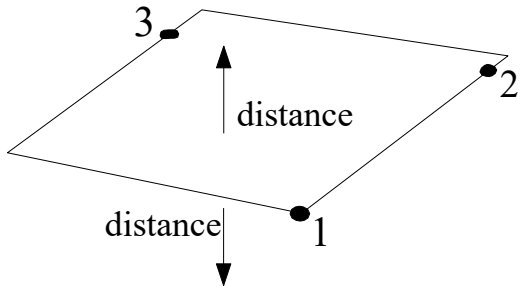


LIMITMOD type x1 y1 z1 x2 y2 z2 x3 y3 z3 dist.

Parameter	*Description*
type	Description type: Type = 2 : Elements closer to plane than 'dist.' are included Type = -2 : Elements closer to plane than 'dist.' are excluded
x1, y1, z1	Coordinates of point 1 used to define the limitation plane
x2, y2, z2	Coordinates of point 2 used to define the limitation plane
x3, y3, z3	Coordinates of point 3 used to define the limitation plane
dist.	Distance from plane See figure below. Default: Entire structure included.

With this record, the user may extract the part of the entire structure, which is of interest with respect on temperature distribution. All elements closer the defined plane (type = 2) or at larger distance from the plane (type = -2) will be included in the transient heat transfer analysis, and *BELTEMP* records will be generated for each of these elements. Elements, which do not fulfil the limitation-test are disregarded, (and are visualized).

This record may be repeated.



LIMITMOD type iel1 iel2 iel3 .....

Parameter	*Description*
type	Description type: Type = 3 : Specified elements only are included Type = -3 : Specified elements are excluded
Iel1, iel2 ...	Specification of the elements to be included/excluded

With this record, the user may extract the elements, element-by-element included/excluded in the transient heat transfer analysis.

NOTE! Type = 3 is not available together with other LIMITMOD types, and all elements have to be specified in the same LIMITMOD record.

Type = -3 may be used together with the other LIMITMOD types.

LIMITMOD 3 is given once!
LIMITMOD -3 may be repeated.

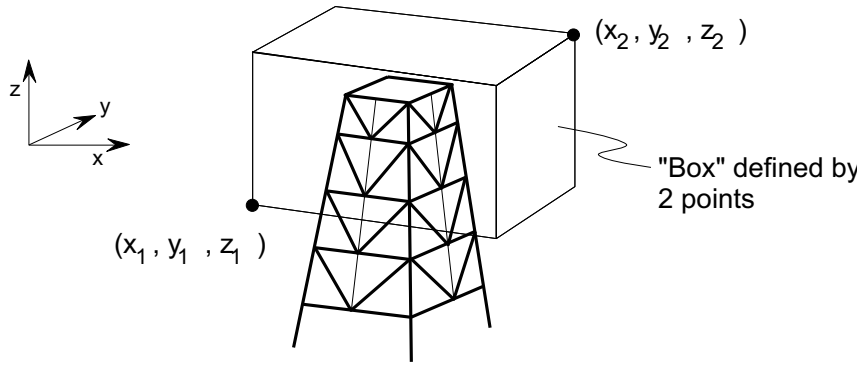
Limit Elements exposed to fire

LIMTFIRE type x1 y1 z1 x2 y2 z2	
Parameter	*Description*
type	Description type: Type = 1 : Elements inside box are exposed to fire Type = -1 : Elements outside box are exposed to fire
x1, y1, z1 x2, y2, z2	Coordinates of point 1 used to define the limitation "box". Coordinates of point 2 used to define the limitation "box". See figure below. Default: All elements are exposed to fire.

With this record, the user may extract the part of the entire structure, which will be exposed to fire. All elements inside, (type = 1) or outside (type = -1) the defined "box" will be exposed to the actual fire scenario. However, heat transfer calculations will be calculated for all elements fulfilling the *LIMITMOD* limitations.

This record may be repeated.

Note! This option is dummy if the fire scenario is calculated by KFX.



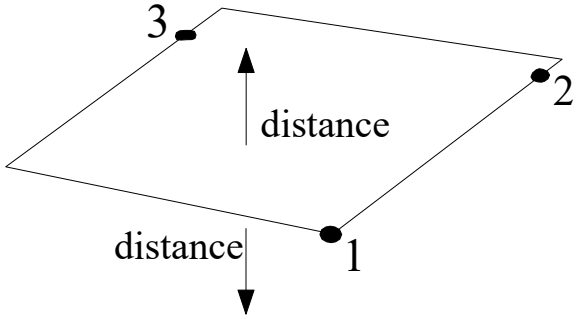
LIMTFIRE type x1 y1 z1 x2 y2 z2 x3 y3 z3 dist.

Parameter	*Description*
type	Description type: Type = 2 : Elements closer to the plane than 'dist.' are exposed to fire Type = -2 : Elements outside plane are exposed to fire
x1, y1, z1	Coordinates of point 1 used to define the plane.
x2, y2, z2	Coordinates of point 2 used to define the plane.
x3, y3, z3	Coordinates of point 3 used to define the plane.
Dist.	Distance from plane See figure below Default: All elements are exposed to fire.

With this record, the user may extract the part of the entire structure, which will be exposed to fire. All elements closer to the plane than "dist.", (type = 2) or at larger distance from the plane (type = -2), will be exposed to the actual fire scenario. However, heat transfer calculations will be calculated for all elements fulfilling the *LIMITMOD* limitations.

This record may be repeated.

Note! This option is dummy if the fire scenario is calculated by KFX.



LIMTFIRE type iel1 iel2 iel3 ....

Parameter	*Description*
type	Description type: Type = 3 : Specified elements are exposed to fire Type = -3 : Specified elements are not exposed to fire
iel1, iel2, ...	Actual elements to be exposed/not exposed to fire. Default: All elements are exposed to fire.

With this record, the user may specify the elements, which will be exposed to fire, (type = 3) or not exposed to fire, (type = -3).

This record may be repeated.

Note! This option is dummy if the fire scenario is calculated by KFX.

Temperature Dependency

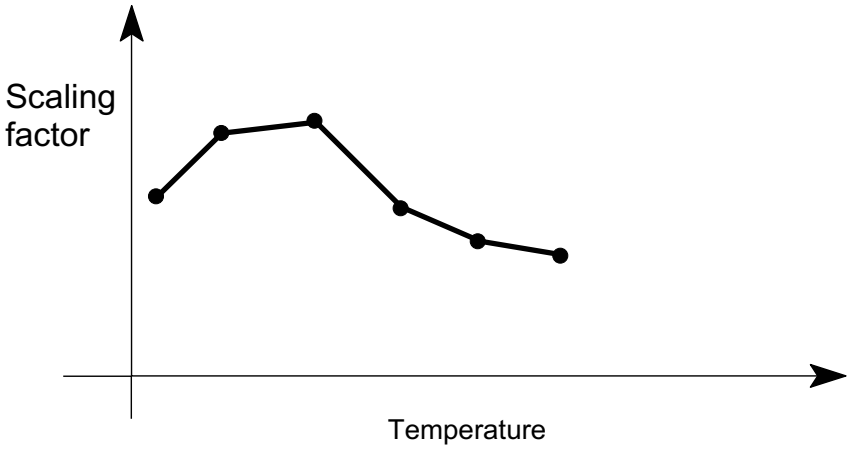
TEMPDEPY dep_no temp1 factor1 temp2 factor3 ...

Parameter	*Description*
dep_no	Temp dependency number (used defined ID)
temp1	temp (Celsius degrees)
factor1	Scale factor 1
temp2	temp (Celsius degrees)
factor2	Scale factor 2

With this record, the user specifies a temperature dependency by discrete points (maximum 25 points).

Values between the tabulated points are interpolated, and values outside the specified range are extrapolated as indicated in the figure below.

This record may be repeated.



THERMDEP therm_ID Dep_rho Dep_c Dep_k Dep_e

Parameter	*Description*
Therm_ID	Thermal property ID to be connected to temperature dependency definitions.
Dep_rho	Mass density is scaled according to the specified curve. 0 means temp indep.
Dep_c	Heat capacity is scaled according to the specified curve. 0 means temp indep.
Dep_k	Conductivity is scaled according to the specified curve. 0 means temp indep.
Dep_e	Emissivity is scaled according to the specified curve. 0 means temp indep.

With this record, the user connects temperature dependency curves ("TEMPDEPY") to the different "THERMPAR" parameters.

This record may be repeated.

INSULDEP Insul_ID Dep_k Dep_e

Parameter	*Description*
Insul_ID	Insulation ID to be connected to temperature dependency definitions.
Dep_k	Conductivity is scaled according to the specified curve. 0 means temp indep.
Dep_e	Emissivity is scaled according to the specified curve. 0 means temp indep.

With this record, the user connects temperature dependency curves ("TEMPDEPY") to the different "INSULPAR" parameters.

This record may be repeated.

No Linearization of Temperature Field

NOLINEAR	**Element1** **Element2** **....**
Parameter	*Description*
Element1 Element2	The calculated temperatures at the specified elements (external ID) are **not** linearized and no *BELTEMP* records are written for the elements. Instead, the *ELEMTEMP* records are written, and the temperature fields calculated by FAHTS are written to file without any loss of information due to linearization. **NOTE!** If no elements are specified, *ELEMTEMP* records are written for **all** elements.

With this record, the user specifies the elements with no linearization of the temperature field.

The detailed temperature field is necessary if **local behaviour** should be accounted for in the mechanical response analysis.

This record may be repeated.

Linearization of Temperature Field

LIN_RULE ***Rule ListType Id-list***

Parameter	*Description*	*Default*
Rule	Actual Rule to be used 0 : Linearized temperatures based on virtual work principle. 1 : Mean temp.=volume average, gradients=0 2 : Temperature plane from regression. **Max** temperature section is used. 3 : Max Temperature, Gradients=0 -2 : Temperature plane from regression. **Min** temperature section is used.	0
ListType	Definition of the ID-list: *Element* : Element-ID list *Material* : Material-ID list : *Elements referring to the materials*. *Geometry* : Geometry-ID list : *Elements referring to the geometries*. *Group* : Group-ID list : *Elements in the groups*. *All* : All elements	All

With this record, the user specifies the linearization rule of the temperature field for the specified elements. Assignment to

This record could be repeated.

Example:

```
Lin_Rule  2              ! Define to all
Lin_Rule -2  Mat 11 101 ! Use minimum temp (cryogenic spills)for
                         ! specified materials
```

Temperature plot of specified points

TEMP_PLO	***X1***	***Y1***	***Z1***
	X2	***Y2***	***Z2***
	…..	***….***	***….***
	Xn	***Yn***	***Zn***

Parameter	*Description*
X1, Y1, Z1	Coordinates of 1'st point.
X2, Y2, Z2	Coordinates of 2'nd point

With this record, the user specifies coordinates for points in the temperature FE model to be printed as function of analysis time to a separate plot file (named: *<prefix>_temp.plo*. The *nearest* nodal point in the FAHTS temperature model is used, and both input coordinates and the coordinates of the nearest point are printed in the file header.

This record is specified only once.

PARTLY Exposed / Insulated Members

EXP_ELEM GeoTyp { Exp_Codes} {PFP_Codes}. Elm1 Elm2…...

Parameter	*Description*
GeoTyp	Cross Section Type: GeoTyp = 2 : I-Profile GeoTyp = 3 : Box
Exp_Codes / PFP_Codes	The Codes are :0,1, 2 or 3. 0 : No faces are exposed / protected 1 : Surface facing *outwards* is exposed / protected 2 : Surface facing *inwards* is exposed / protected 3 : Both Surfaces are exposed / protected Depending of GeoTyp, 3 or 4 codes are required, and are defined as follows : GeoTyp = 2 : Top Flange Web Lower Flange GeoTyp = 3 : Top Flange Left Side Right Side Lower Flange .
elm1,elm2..	Actual elements to be exposed/protected as defined By default, all surfaces are exposed and (optionally) insulated.

With this record, the user may specify (element by element) the members, which have a non-default heat exposure and PFP cross section cover.

Example:

```
'                          * Exposure  *     * Passive Fire Protection *
'            GeoTyp        top  web  bott          top  web  bott       Elm1      Elm2
EXP_ELEM       2            1    0    0             2    3    3         1020      2020
```

In this case, Elements 1020 and 2020 (which are I-Profiles) are exposed on the *over* side of the upper flange only. The profile is fully protected, except for the upper flange, which is protected on the *under*side only.

NOTE
EXP_ELEM will override previous definition (f ex using EXP_IPRO, EXP_BOX etc.), and makes it possible to define typical protection/PFP (using the profile definition) and then specify element by element the members differing from the typical.

This record may be repeated.

EXP_IPRO { Exp_Codes} {PFP_Codes}. Geo1 Geo2…...

Parameter	*Description*
Exp_Codes / PFP_Codes	Codes for : *Top Flange* *Web* *Lower Flange* The Codes are :0,1, 2 or 3. 0 : No faces are exposed / protected 1 : Surface facing *outwards* is exposed / protected 2 : Surface facing *inwards* is exposed / protected 3 : Both Surfaces are exposed / protected .
Geo1,Geo2	Actual Cross Section(s) to be exposed/protected as defined By default, all surfaces are exposed and (optionally) insulated.

With this record, the user may specify the cross sections, which have a non-default heat exposure and PFP cross-section cover.

Example:

```
'              * Exposure  *     * Passive Fire Protection *
'              top  web  bott          top  web  bott       Geo1     Geo22
EXP_IPRO       1    0    0             2    3    3          24012    32015
```

In this case, all elements referring to either geo 24012 or 32015 are exposed on the *over* side of the upper flange only. The profile is fully protected, except for the upper flange, which is protected on the *under* side only.

NOTE : If **no** geometry IDs are specified, **all I profiles** get the specified exposure/ PFP cover

This record may be repeated.

EXP_BOX { Exp_Codes} {PFP_Codes}. Geo1 Geo2…...

Parameter	*Description*
Exp_Codes / PFP_Codes	Codes for : Top Flange Left Side Right-side Lower Flange Because the Box is a hollow section, the Codes are only: 0 or 1. 0 : Surface is ***not*** exposed / protected 1 : Surface is exposed / protected .
Geo1,Geo2	Actual Cross Section(s) to be exposed/protected as defined By default, all surfaces are exposed and (optionally) insulated.

With this record, the user may specify the cross sections, which have a non-default heat exposure and PFP cross section cover.

Example:

```
'                 * Exposure  *                 * Passive Fire Protection *
'               top Lside Rside  bott               top Lside  Rside  bott        Geo1     Geo22
EXP_BOX         1     1     1     0                   0    1     1      1          200400   300400
```

In this case, all elements referring to either geo 200400 or 300400 are exposed on the *over* side of the upper flange and on both sides. The profile is fully protected, except for the upper flange.

NOTE : If **no** geometry IDs are specified, **all Box profiles** get the specified exposure/ PFP cover

This record may be repeated.

EXP_QUAD { Exp_Codes} {PFP_Codes}. Geo1 Geo2…...

Parameter	*Description*
ExpCodes/ PFP_Codes	Codes for : the membrane/shell elements The Codes are :0,1, 2 or 3. 0 : No faces are exposed / protected 1 : Surface facing *outwards* is exposed / protected 2 : Surface facing *inwards* is exposed / protected 3 : Both Surfaces are exposed / protected .
Geo1,Geo2	Actual Cross Section(s) to be exposed/protected as defined By default, all surfaces are exposed and (optionally) insulated.

With this record, the user may specify the cross sections, which have a non-default heat exposure and PFP cross section cover.

Example:

```
'              * Exposure *      Passive Fire Protection      Geo1    Geo22
EXP_QUAD          1                        2                  12      15
```

In this case, all membrane/shell elements referring to either geo 12 or 15 are exposed on the *over* side of the upper flange only. The plate protected on the *under*side only.

NOTE : If **no** geometry IDs are specified, **all** quad elements get the specified exposure/ PFP cover

This record may be repeated.

EXP_GRUP GeoTyp { Exp_Codes} {PFP_Codes}. Grp1 Grp2 …...

Parameter	*Description*
GeoTyp	Cross Section Type: GeoTyp = 2 : I-Profile GeoTyp = 3 : Box
ExpCodes/ PFP_Codes	The Codes are :0,1, 2 or 3. 0 : No faces are exposed / protected 1 : Surface facing *outwards* is exposed / protected 2 : Surface facing *inwards* is exposed / protected 3 : Both Surfaces are exposed / protected Depending of GeoTyp, 3 or 4 codes are required, and are defined as follows : GeoTyp = 2 : Top Flange Web Lower Flange GeoTyp = 3 : Top Flange Left Side Right Side Lower Flange
Grp.	Actual groups to be exposed/protected as defined

With this record, the user may specify (group-by-group) the members, which have a non-default heat exposure and PFP.

Example:

```
'                  * Exposure  *  *Passive Fire Protection*
'         GeoTyp   top  web  bott   top  web  bott          Grp1  Grp2
EXP_GRUP    I       1    0    0      2    3    3            11    12
```

In this case, Groups 11 and 12 (which are I-Profiles) are exposed on the *over* side of the upper flange only. The profile is fully protected, except for the upper flange, which is protected on the *under*side only.

NOTE
EXP_GRUP will override previous definition (f ex using EXP_IPRO, EXP_BOX etc.), and makes it possible to define typical protection/PFP (using the profile definition) and then specify group by group the members differing from the typical.

This record may be repeated.

KAMELEXP TimeBegin TimeEnd …...{ListType} { Id List }

Parameter	*Description*	*Default*
TimeBegin	Time to start using data from the KFX database, (the "k2f" files). NOTE! The time is given in same unit as used in the TEMPSIM	0.0
TimeEnd	Time to stop using data from KFX (i.e. starts the cooling)	Inf
ListType	How to define the actual elements Elem : Elements are specified directly (Element ID's) Mat : Elements are identified through their material ID Geo : Elements are identified through their geometry ID Group : Elements are identified through groups	
Id list	List of actual IDs.	

With this record, the user may specify KFX exposure-times (begin/end) on different structural elements.

Example:

```
'
'          TimeBegin  TimeEnd   ListType   Grp1  Grp2
KAMELEXP       0.0     10        Group      11    12
```

This means that element groups 11 and 12 receive heating from KFX between time = 0 and 10. After 10 (minutes), the elements receive no heating.

This record may be repeated.

NOEXPOSURE {ListType} { Id List }

Parameter	*Description*
ListType	How to define the actual elements Elem : Elements are specified directly (Element ID's) Mat : Elements are identified through their material ID Geo : Elements are identified through their geometry ID Group : Elements are identified through groups From2Inc : Elements are listed: First ID Last ID Increment Ge.and.Ma : Elements with both Geometry and Material are identified Ge.not.Ma : Elements with Geometry but not Material are identified Ma.not.Ge : Elements with Material but not Geometry .
Id-List	Actual ID's By default, all surfaces are exposed

With this record, the user may specify the elements, which should ***not*** be exposed to radiation from the **KAMELEON** (KFX)fire source. Internal radiation, however. will still become active.

Example 1:

```
'             * ListType    Id-List
NoExposure      Mat       12   15      1012      1015
```

Elements with materials 12, 15, 1012, 1015 will not receive radiation from the KAMELEON fire.

Example 2:

```
'             *   ListType      MatID    GeoID
NoExposure      Ma.and.Ge     12       20010
```

Elements with the property combination Material ID 12 and Geometry ID 20010, will not receive radiation from the KAMELEON fire.

Example 3:

```
'             *   ListType      MatID    GeoID
NoExposure      Ma.not.Ge     12       20010
```

Elements with Material ID 12 and Geometry ID different from 20010, will not receive radiation from the KAMELEON fire.

This record may be repeated.

PFPSCALE CrossSect Part Value Geo1 Geo2…...

Parameter	*Description*
CrossSect	Cross section shape: • IProfile • BOX
Part	Cross Section Part • Web • UpFlange • LoFlange
Value	Actual Scaling parameter. The U-value of the actual part becomes U x Value.
Geo1,Geo2	Actual Cross Section(s) with scaled U-value

With this record, the user may specify the cross sections with uneven U-Value for the different section parts.

This record may be repeated.

Example:

'	** CrossSect **	*Part*	*Value*	*Geo1*	*Geo2*
PFPScale	*IProfile*	*Web*	*0.5*	*12*	*15*

In this case, the U-Value for the web for geo 12 or 15 becomes 0.5 times the U-Value given under INSULPAR.

EXPLOCAL PropID CrossSect Operation Part Min Max (define)
--- " ---
--- " ---
Property
ListType ID-List (assign)

Parameter	*Description*
PropID	Property ID
CrossSect	Cross section shape: IProfile BOX
Operation	Actual Operation: SizeArea : Define the size of the local area Property : Definition of local property: Exposed or Unexposed. Assign : Assign properties to elements
Part	Cross Section Part Web (L_side and R_side for box) UpFlange LoFlange Length
Min Max	Definition of min and max area of the actual elements: Length : Dimensionless coordinate X/L_{MIN} X/L_{MAX} Else : Section coordinates
ListType ID-List	How to define the actual elements: Elem, Mat, Geo or Group. List of IDs.
Property	Property of the actual area: NoExposure : The actual area is unexposed.

With this record, the user may specify that only a small area of an element is exposure.
This record may be repeated.

Example:

```
'            ID    Cross     Oper       Part       Min    Max
 ExpLocal   100    IProf   SizeArea    Web        0.060   0.100
 ExpLocal   100    IProf   SizeArea    U_flange  -1.000   1.000
 ExpLocal   100    IProf   SizeArea    Length     0.25    0.5
'
 ExpLocal   100    IProf   Property    NoExposure
'
 ExpLocal   100    IProf   Assign      Elem  10  11  12
```

In this case, local area of on Web and Upper Flange, for x/L between 0.25 and 0.5, the element surface is unexposed. This property is assigned to elements: 10, 11 and 12.

PFPLOCAL	**PropID**	**CrossSect**	**Operation**	Part	Min	Max	(define)
	- - - " - - -			Property			
	- - - " - - -			ListType	ID-List		(assign)

Parameter	*Description*
PropID	Property ID
CrossSect	Cross section shape: IProfile BOX
Operation	Actual Operation: SizeArea : Define the size of the local area Property : Definition of local property: Exposed or Unexposed. Assign : Assign properties to elements
Part	Cross Section Part Web (L_side and R_side for box) UpFlange LoFlange Length
Min Max	Definition of min and max area of the actual elements: Length : Dimensionless coordinate X/L $_{MIN}$ X/L $_{MAX}$ Else : Section coordinates
ListType ID-List	How to define the actual elements: Elem, Mat, Geo or Group. List of IDs.
Property	Property of the actual area: noPFP : The actual area is unprotected PFP : The actual area has PFP

With this record, the user may specify opening in PFP. Or, alternatively that only a limited are has PFP. This record may be repeated.

Example:

```
'            ID    Cross     Oper       Part       Min    Max
 PfpLocal   100    IProf   SizeArea    Web        0.060   0.100
 PfpLocal   100    IProf   SizeArea    U_flange  -1.000   1.000
 PfpLocal   100    IProf   SizeArea    Length     0.25    0.5
'
 ExpLocal   100    IProf   Property    noPFP
'
 ExpLocal   100    IProf   Assign      Elem  10  11  12
```

In this case, local area of on Web and Upper Flange, for x/L between 0.25 and 0.5, the surface of the protected element is unprotected. This property is assigned to elements: 10, 11 and 12.

NOTE: The element has to be defined with PFP (ELMINSUL or INS_GRUP) in addition.

MISCELLANEOUS

FatGroup	
Parameter	*Description*
With this record, the user may switch ON visualization of groups in the graphical user interface, (xact). This record is given once.	

PushDown	
Parameter	*Description*
With this record, the user may switch ON preparation for "PushDown" analysis in USFOS. The option means that additional temperature data are stored on the "beltemp" file. The option is recommended for cases with extreme gradients (for example when partly exposed or partly protected elements exist in the model). This record is given once.	

SWITCHES	***Keyword***	*SubKey*	Value	(value2)

Parameter	*Description*	*Default*
KeyWord	Definition of actual switch / parameter: **FahtsOpt** : FAHTS specific switches	
SubKey	Specification of actual FAHTS parameter to set	
Value	Parameter value	

With this record the user may override the boundary conditions defined on the structural file. In the analysis the specified boundary codes are applied to all listed nodes. The nodes are listed directly (node ID list) or all in once.

Examples *:*

```
SWITCHES FahtsOpt Cryogen ON -160 ! Accept temperatures to -160 in connection
                                  ! with linearization.
```

This record may be repeated.

See below for complete list of SWITCHES options.

KeyWord	SubKey	Value	*Val_2*	Description	Default
FahtsOpt	*Cryogen*	ON	(TempLim)	Switch ON handling of low temperatures, (below zero). The low temperature limit could be set.	OFF
	FluxDelay	Delay	N/A	Smooth transition between one KFX-fire to the next.	0.0
	NewEnergy	ON/OFF	N/A	Using new or "old" energy module	ON
	ShieldViz	ON/OFF	N/A	Visualization of Shield	ON
	PriIniTemp	ON/OFF	N/A	Print of initial temperatures on the beltemp file.	OFF
	WriNodeTmp	Abaqus	(MinTemp)	Print node temperatures above "MinTemp" to ABAQUS If MinTemp is not given, 20°C is used.	OFF

SHIELD ShieldFact x1 y1 z1 x2 y2 z2 x3 y3 z3

Parameter	*Description*
ShieldFact	Shielding factor: 0.0 : No shielding 1.0 : Fully blocked
x1, y1, z1	Coordinates of point 1 used to define the plane.
x2, y2, z2	Coordinates of point 2 used to define the plane.
x3, y3, z3	Coordinates of point 3 used to define the plane.

With this record, the user may define radiation shields, which will limit the radiation from the user defined heat sources (USERFIRE, USERJET and USERFLARE).

This record may be repeated.

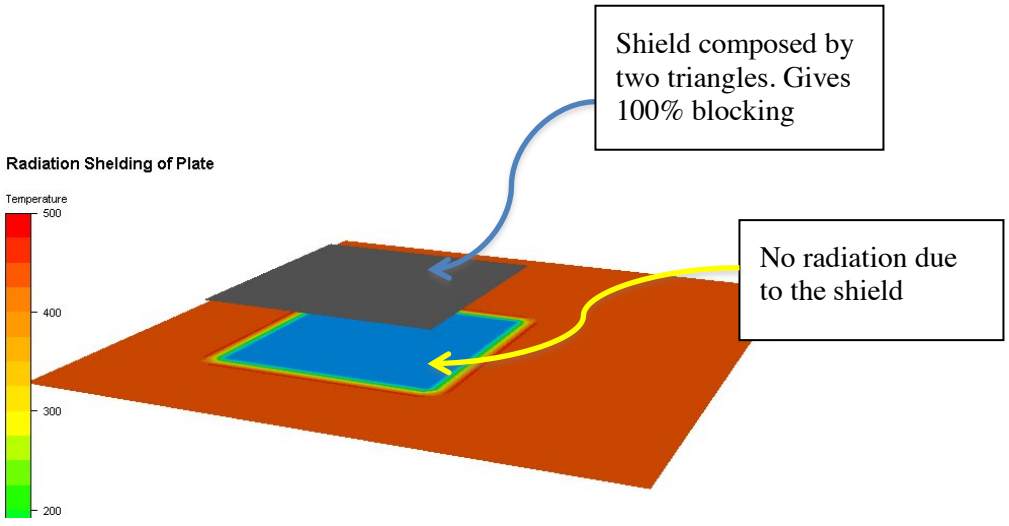
NOTE! This option is dummy if the fire scenario is calculated by KFX.

Example:

```
'          ShieldFact    X1 Y1 Z1    X2 Y2 Z2    X3 Y3 Z3
 SHIELD        1.0       2  2  1     6  2  1     6  6  1
 SHIELD        1.0       2  2  1     2  6  1     6  6  1

'          history  x    y    z      ref_intensity ref_radius
 USERFIRE      0    5    0   10.0E3     40E3         10.0E3     ! Parallel rays.
```

Gives a rectangular shield as shown on the image. The parallel rays from above (40 kW/m²) are 100% blocked by the shield.



ReMesh_I

Parameter	*Description*
With this record, the user switch ON re-meshing of I-profiles. Elements with even heat exposure and even PFP do not need a fine mesh. However, if FAHTS detects either uneven exposure or PFP (i.e. the user has used the EXP_ commands) on I-sections the mesh density increases automatically. (3 elements for web and 4 on the flanges). This record is given once.	

ReMesh_B

Parameter	*Description*
With this record, the user switch ON re-meshing of BOX-profiles. Elements with even heat exposure and even PFP do not need a fine mesh. However, if FAHTS detects either uneven exposure or PFP (i.e. the user has used the EXP_ commands) on BOX-sections the mesh density increases automatically. (3 elements for top/botm and the sides). This record is given once.	