

4 EXAMPLES

1. INTRODUCTION	2
1.1 General information	2
1.2 Basic commands	6
2. EXAMPLES OF USE	9
2.1 Single beam exposed to fire and protected on four sides.	9
2.2 Single beam exposed to fire on three sides.	13
2.3 Single beam exposed to fire on one side.	15
2.4 Single beam exposed to fire and protected on three sides.	16
2.5 Single beam exposed to fire on three sides.	17
2.6 Beam with displacement limitation and temp. dependant material properties.	19
2.7 Cantilever beam fully exposed to fire.	21
2.8 Cantilever beam exposed to two radiation sources.	25
2.9 Simple portal frame.	27
2.10 Simple portal frame.	30
2.11 3 Bay and 3 Storey frame.	33
2.12 9 Bay and 3 Storey frame.	37
2.13 Arched framework exposed to radiation and heat increment.	41
2.14 Single frame exposed to pool fire placed between columns.	44
2.15 Single frame exposed to pool fire placed between columns.	48

Summary of the examples

	Mat.	Prof.	Exposure	Protection	Options1	Options2
Ex. 1:	Alum	Box	Fully exp.	Fully prot.	Fire exp. <i>a)</i>	Protection
Ex. 2:	Alum	Box	Partly exp.	Fully prot.	Fire exp. <i>b)</i>	Protection
Ex. 3:	Alum	Box	Partly exp.	Fully prot.	Fire exp. <i>c)</i>	Protection
Ex. 4:	Alum	Box	Partly exp.	Partly prot.	Fire exp. <i>d)</i>	Protection
Ex. 5:	Alum	I-prof	Partly exp.	Partly prot.	Fire exp. <i>e)</i>	Protection
Ex. 6:	Alum	I-prof	Partly exp.	Partly prot.	Displ. lim.	Temp. dep
Ex. 7:	Steel	Pipe	Partly exp.	Unprotected	User source	Radiation
Ex. 8:	Steel	Pipe	Partly exp.	Unprotected	User source	Radiation
Ex. 9:	Steel	I-prof	Fully exp.	Unprotected	User source	Temp. hist.
Ex. 10:	Steel	I-prof	Fully exp.	Unprotected	User source	Temp. hist.
Ex. 11:	Steel	I-prof	Partly exp.	Unprotected	User source	Temp. hist.
Ex. 12:	Steel	I-prof	Partly exp.	Unprotected	User source	Temp. hist.
Ex. 13:	Steel	I-prof	Partly exp.	Unprotected	Radiation	Temp. hist.
Ex. 14:	Steel	Pipe	Fully exp.	Unprotected	Kameleon	Static ana.
Ex. 15:	Steel	Pipe	Fully exp.	Unprotected	Kameleon	Temp. sim.

1. Introduction

1.1 General information

This manual contains some examples that describe the method for fire analysis by use of FAHTS and USFOS.

FAHTS and USFOS are interlinked. Both are needed for a fire analysis of a structure. The output from FAHTS is input for USFOS.

The FAHTS input can be given in one or two files. In this manual all input are separated in a *Control file* and a *Structure file*. (See Figure 1.1-1 for a *Structure file*.)

The input contains information about the heat source, thermal parameters for the structure and insulation, the structure with node numbers, co-ordinates, elements, cross-section geometry and mechanical loads. It is not necessary to give commands with default values if the default value is suitable.

The output includes an out file Figure 1.1-2(short version), which contains a summary of the input and the temperature analysis done by FAHTS. The results include Time, Accumulated energy, Max heat input and Max/Min temperature after each load step. Another output is the *Beltemp file*, Figure 1.1-3, which includes the temperature increment of the elements for each load step of the fire load.

Some of the files shown in this introduction chapter are not included in the examples. This is because they take up too much space, and because they have no importance to this manual.

Filenames used in the examples:

<u>File type:</u>	<u>File name:</u>
FAHTS Control file	fahts.fem
USFOS Control file	usfos.fem
FAHTS out file	resf.out
USFOS out file	resu.out
Structure file	stru.fem

```

HEAD          Structure file

          Single beam
-----
          ELEMENT AND NODE NUMBERS
-----
          * Node co-ordinates. Boundary conditions *
          Node ID      X      Y      Z      Boundary code
          x  y  z  rx  ry  rz
NODE          1      0.000  0.000  0.0      1  1  1  0  0
NODE          2      2.200  0.000  0.0      0  1  1  0  0
NODE          3      4.400  0.000  0.0      0  1  1  0  0
-----
          * Element numbers and node numbers. Geometry ID *
          Elem_ID  np1  np2  material  geom  lcoor  ecc1  ecc2
BEAM          12      1      2      1      10
BEAM          23      2      3      1      10
-----
          CROSS-SECTION GEOMETRY
-----
          Geom_ID  H      T-side  T-bott  T-top  Width
BOX           10      0.300  0.008  0.008  0.008  0.200
-----
          LOAD CASES
-----
          Load_case Node_ID  Fx      Fy      Fz
NODELOAD      1          2      0      0      -1.0E4
-----
          Load_case  AccX  AccY  AccZ
GRAVITY        2          0      0     -9.81

```

Figure 1.1-1: Structure file for FAHTS and USFOS, *stru.fem*.

```

***** F A H T S *****
*****
===== A N A L Y S I S   P A R A M E T E R S   =====

          F A H T S temperature simulations
          Single beam, box cross-section
          Example No. 1

          Number of nodal points      =      3
          Number of elements          =      2
          +++++
          FAHTS
          Step      Time      Accum.      Max heat      Max      Min
          no.      (s)      energy      input      temp.      temp.
                        (J)      (W/m2)      (degr C)      (degr C)
          1      36.00  6.338E+04  1.13E+03      16.0      14.9
          2      72.00  2.972E+05  2.62E+03      19.4      15.3
          3     108.00  7.562E+05  4.69E+03      26.2      16.5
          4     144.00  1.455E+06  6.45E+03      36.4      19.1
          5     180.00  2.371E+06  8.18E+03      49.9      23.3
          6     216.00  3.405E+06  8.39E+03      65.0      29.2
          7     252.00  4.441E+06  8.29E+03      80.0      36.5
          8     288.00  5.478E+06  8.49E+03      94.8      44.9
          9     324.00  6.515E+06  8.38E+03     109.4      53.9
          10     360.00  7.548E+06  8.57E+03     123.8      63.2
          11     396.00  8.579E+06  8.45E+03     138.0      72.7
          12     432.00  9.609E+06  8.64E+03     151.9      82.5
          13     468.00  1.064E+07  8.51E+03     165.6      92.8
          14     504.00  1.165E+07  8.39E+03     178.9     103.3
          15     540.00  1.266E+07  8.56E+03     191.9     113.7
          16     576.00  1.366E+07  8.44E+03     204.8     124.0
          17     612.00  1.464E+07  8.31E+03     217.3     134.4

```

Figure 1.1-2: Out file from FAHTS analysis, *resf.out*.

```

*****
** Temperature increments **
** calculated **
** by **
** F A H T S **
*****
*****
load case      time
LCASETIM      4      360.0

load_case      element      Mean      Gradient      Gradient
no            number      temperature      Y-dir      Z-dir
(increment)  (increment)  (increment)
BELTEMP      4      12      103.722      -.088      -171.821
BELTEMP      4      23      103.722      -.088      -171.818

load_case      time
LCASETIM      5      720.0

load_case      element      Mean      Gradient      Gradient
no            number      temperature      Y-dir      Z-dir
(increment)  (increment)  (increment)
BELTEMP      5      12      117.532      -.274      -79.993
BELTEMP      5      23      117.532      -.274      -79.984

load_case      time
LCASETIM      6      1080.0

load_case      element      Mean      Gradient      Gradient
no            number      temperature      Y-dir      Z-dir
(increment)  (increment)  (increment)
BELTEMP      6      12      100.828      -.516      -9.241
BELTEMP      6      23      100.827      -.516      -9.225

```

Figure 1.1-3: Beltemp file from FAHTS analysis, *resf_beltemp.fem*.

The *Beltemp* file from FAHTS is one of the input files for USFOS. Other input for USFOS is in this manual separated in two files, a *Control* file and a *Structure* file. Before running USFOS it can be smart checking out file and inspecting in XFOS. USFOS gives a *Status* file, an out file and a raf-file. The *Status* file (see Figure 1.1-4) contains a short summary of the results from USFOS. Among other things, time before yield and plastic hinge, and utilization of the beam. The USFOS .out file contains a summary of the input and a list of displacement and reaction forces for each load step. The raf-file contains all the information needed for a view in XFOS. This is not a legible file.

```

      --- U S F O S   A N A L Y S I S   S T A T U S ---

      U S F O S progressive collapse analysis
      Single beam, box cross-section
      Example No. 1

      Input files:
                        usfos.fem
                        stru.fem
                        res_beltemp.fem

      Number of nodal points      :      3
      Number of elements         :      2

                                     Time
      First yield at              :      1146.413
      First plastic hinge at      :      1206.374
      No buckling detected
      Exceeding utilization threshold of :      1.10 at :      1207.094
      Max number of negative pivot element :      1 at :      1207.094
      Analysis terminated at      :      1207.094

      --      Yielding Status      --
      Elem ID  Position  Cross sect.      Time
                  utilization
           12     node 2      .83          1146.413
           13     node 1      .83          1146.413

      --      Plastic hinge Status  --
      Elem ID  Position  Cross sect.      Time
                  utilization
           12     node 2      1.00          1206.374
           23     node 1      1.00          1206.734

      -- Elements Exceeding utilization of  1.10 --
      Elem ID  Position  Cross sect.      Time
                  utilization
           23     node 1      1.11          1207.094

      ----- E N D   A N A L Y S I S   S T A T U S -----

```

Figure 1.1-4: Status-file from USFOS, *resu_status.text*.

It's important to write input files in an orderly way, to obtain a good readability. This makes it easier to find errors and correct them, and it's easier to see what the file contains.

Space is used as the only dividing sign. The number of spaces is of no importance.

It's therefore possible to line up the numbers under each other.

Please note that the program reads character by character, and a command lasts until a new command is given.

It is possible to write explanations to the commands and make "headings" as long as one of the comment characters are put in front. See USFOS User's Manual. The program will ignore the text behind these characters. The command characters used in the examples in this manual are mostly " " and " ! ".

1.2 Basic commands

This chapter contains information about some of the basic FAHTS commands.

The commands are also explained in the FAHTS User's Manual.

It is very important to write a command only once, if it's said so in the manual. If a command is given more than one time, and are changed a bit, this can lead to an analysis error.

HEAD:

This command is written in the FAHTS and USFOS input files. It makes room for three lines of text identifying the analysis. The three lines will be repeated in some of the output files.

<pre> HEAD U S F O S progressive collapse analysis Single beam HUP </pre>
--

Figure 1.2-1: HEAD command.

MOVIEPRI:

By writing this command the storing of the temperature analysis model and the temperature result at the specified time increments are switched **ON**. This makes it possible to inspect the structure by XFOS before running USFOS.

SHAPFACT:

By writing this command shape factor calculations are switched ON. Radiation between temperature elements belonging to the same FEM structural element is included. (Radiation inside tubes).

TEMPMESH:

Specifies the temperature mesh refinement of the profile. Note that this command is overridden by the MESH... commands. See FAHTS User's Manual for details.

<pre> ' n_length n_circ TEMPMESH 2 16 </pre>
--

Figure 1.2-2: TEMPMESH command.

MESHBOX:

Specifies the temperature mesh refinement for box cross-sections. See FAHTS User's Manual for details.

<pre> '----- ' PROFILE MESH '----- ' n_length n_side n_top n_bott MESHBOX 2 8 4 4 </pre>

Figure 1.2-3: MESHBOX command.

MESHIPRO:

Specifies the temperature mesh refinement for I cross-sections. See FAHTS User's Manual for details.

----- PROFILE MESH -----							
	n_length	n_web	n_top	n_bott	elem_id	elem_id	elem_id
MESHIPRO	8	1	2	2			
MESHIPRO	8	6	6	6	1112	111	212

Figure 1.2-4: MESHIPRO command.

MESHPIPE:

Specifies the temperature mesh refinement for pipe cross-sections. See FAHTS User's Manual for details.

There are six possible ways to define a thermal "load". These are:

FIREINT:

Specifies that the environmental temperature and radiation are read from the FIREINT result database. See examples with exposure set to FIREINT.

HCFIRE:

Specifies that the environmental temperature field follows the standard hydrocarbon fire temperature development defined by NPD. See examples with option Fire exposure a), b), c), d) and e).

ISOFIRE:

Specifies that the environmental temperature field follows the standard ISO temperature development, as specified in the ISO-834.

USERFIRE:

The user specifies a radiating source used for simplified fire scenarios, which may be constant (hist = 1), or vary as a function of time, according to the specified time history. See examples with the option Radiation.

USERTEMP:

The user specifies the environmental level, which may be constant (hist = 1), or vary as a function of time, according to the specified time history. See examples with the option Temp.history.

USERJET:

The user specifies a jet source used for simplified fire scenarios, which may be constant (hist = 1), or vary as a function of time, according to the specified time history. See FAHTS User's Manual.

TEMPSIM:

This command **MUST** be supplied.

The command TEMPSIM contains information about the duration of the fire, the number of time steps to be used, and time between saving of results. The last number tells that a new load case is written to file every 900 sec (in Figure 1.2-5). The temperature results are saved to the *Beltemp file*, see Figure 1.1-3.

F A H T S temperature simulation Beam + Column			

ANALYSIS TIME AND TEMPERATURE STEPS			

	end-time (s)	nstep	resinc (s)
TEMPSIM	10800.0	300	900.0

Figure 1.2-5: TEMPSIM command.

THERMPAR:

This command MUST be supplied.

Specifies the thermal properties of the materials used in the structure. See figure below.

INSIDPAR:

Specifies the thermal properties of the material inside hollow members. See figure below.

INSULPAR:

Specifies the thermal properties of the insulation used to protect structural members. See figure below.

ELMINSUL:

Specifies the elements, which should have the actual insulation. See figure below.

```

-----
THE MATERIAL'S AND INSULATION'S THERMAL PARAMETERS
-----

* Thermal parameters of the material *
id rho (kg/m3) c (J/kgK) k (W/mK) emiss
THERMPAR 1 2700.0 896.0 200.0 0.20

* Thermal parameters of the inside material *
id rho (kg/m3) c (J/kgK)
INSIDPAR 1 1.273 0.7

* Thermal parameters of the insulation *
insul-ID type K (W/m2K) emiss
INSULPAR 1 1.0 10.0 0.8

* Elements with insulation *
insul_ID Elem1 Elem2 ....
ELMINSUL 1

```

Figure 1.2-6: Thermal parameters.

2. Examples of use

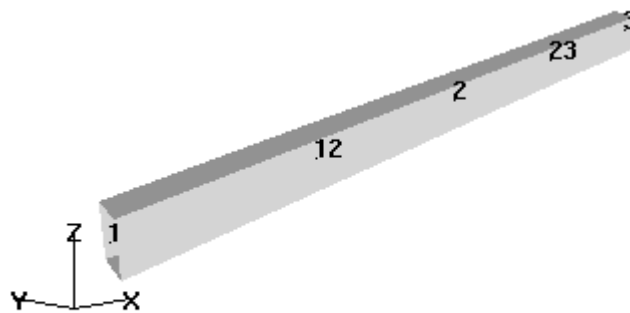
2.1 Single beam exposed to fire and protected on four sides.

Key words:

Ex. 1: Alum Box Fully exp. Fully prot. Fire exp. a) Protection

Material: Aluminium AA 6082-T6. Property reduction due to increased temperature, according to Eurocode 1999 Part 1-2.
Structure: Single beam, aluminium, mid span force, full exposure, full protection.

This example shows the definition of the exposure and the protection of a profile. The beam is modelled with two elements, as shown in Figure 2.1-1.



Analysis model; Time=0

Figure 2.1-1: Structural model with node and element numbers.

Beam length	:	4.400 m	
Profile width	:	0.200 m (width)	
height	:	0.300 m (height)	
thickness	:	0.008 m (thickness)	
Yield stress	:	$\sigma = 255 \text{ MPa}$	Aluminium AA 6082-T6
Youngs modulus	:	$E = 70000 \text{ MPa}$	Aluminium
Mid span force	:	100 kN	
Fire load	:	HC-fire	

Similar beams are used in the three following examples, but with different fire exposure and protection. The *Control files* are therefore only included in their totality in this example. In the following examples only the parts of the *Control files* that are changed are included.

Problem:

Beam with full exposure, and protection on all sides.

Solution:

No specification of exposure and protection, means that the entire cross-section is exposed to the heat source, and protected. This is to simplify the writing of the *Control file*.

```

HEAD          F A H T S temperature simulations
              Single beam, box cross-section

LIN RULE      2
SHAPFACT      ! Calculation of section factor is on.
MOVIEPRI      ! Results from FAHTS can be shown in XFOS.
-----
'             TYPE OF FIRE AND EXPOSURE
'-----
HCFIRE        ! Hydrocarbon fire.
'
'             * Initial temperature in the fire compartment *
'             initemp
INITEMP       15.0
'
'             * Exposure time *
'             end-time (s)  nstep    resinc (s)
TEMPSIM       3600         100      360
-----
'             THE MATERIAL'S AND INSULATION'S THERMAL PARAMETERS
'-----
'             * Thermal parameters of the material *
'             id rho (kg/m3)  c (J/kgK)  k (W/mK)  emiss
THERMPAR      1    2700.0    896.0    200.0    0.20
'
'             * Thermal parameters of the inside material *
'             id rho (kg/m3)  c (J/kgK)
INSIDPAR      1    1.273     0.7
'
'             * Thermal parameters of the insulation *
'             insul-ID  type  K (W/m2K)  emiss
INSULPAR      1        1.0    10.0     0.8
'
'             * Elements with insulation *
'             insul_ID  Elem1  Elem2  ....
ELMINSUL      1                          ! All elements have insulation.
'-----
'             PROFILE MESH
'-----
'             n_length n_side  n_top  n_bott
MESHBOX       2         8       4       4

```

Figure 2.1-2: FAHTS Control file, *fahts.fem*.

```

HEAD          U S F O S progressive collapse analysis
              Single beam
              HUP
,
DETEROFF
CMAXSTEP      1200 ! Increasing from default 512 to 1200.
,
XFOSFULL      ! All data are saved for view in XFOS.
,
-----
LOAD HISTORY AND DEFORMATION CONTROL
-----
* Load history specification with load and deformation *
nloads  npostp  mxpstp  mxpdis
CICYFOS   10    100      1      1
,
lcase lfact mxld  mxdisp  nstep  minstp
2     1.0  1.0   0.0     10   0.001 ! Gravity.
1     1.0  2.0  -0.250  100   0.001 ! Node load.
4     0.02 1.0  -0.250  200   0.001 ! Fahts Fire Load
5     0.02 1.0  -0.250  200   0.001 ! Fahts Fire Load
6     0.02 1.0  -0.250  200   0.001 ! Fahts Fire Load
7     0.02 1.0  -0.250  200   0.001 ! Fahts Fire Load
8     0.02 1.0  -0.250  200   0.001 ! Fahts Fire Load
,
* Specification for deformation control of the structure *
ncnodes
CNODES      1
,
nodex idof  dfact
2          3    1
,
-----
MATERIAL PROPERTIES
-----
mat_no  E-mod  poiss  yield  density  therm.exp
MISOIEP   1  70000.0E6  0.3  255.0E6  2700.0  2.3E-5
,
-----
REDUCTION CURVES OF MATERIAL PROPERTIES
-----
mat_no  dep_E  dep_Yield
USERTDEP   1    1      2
,
curve_no  Temp  Red  ! Reduction curve of E-modul
TEMPDEPY   1    20  1.0
           50  0.99
           100 0.97
           150 0.93
           250 0.78
           300 0.68
           350 0.54
           400 0.40
           550 0.0
,
curve_no  Temp  Red  ! Reduction curve of yield
TEMPDEPY   2    20  1.0
           100  1.0
           150 0.79
           200 0.65
           250 0.38
           300 0.20
           350 0.11
           550 0.0

```

Figure 2.1-3: USFOS Control file, *usfos.fem*.

```

*****
***** F A H T S *****
*****

Version 3.1 / Release 98-03-09
S I N T E F

- Analysis initiated at -
  98-06-15  15:48:42
Licenced to : SINTEF

===== A N A L Y S I S   P A R A M E T E R S   =====

F A H T S temperature simulations
Single beam, box cross-section

Number of nodal points      =      3
Number of elements          =      2
Minimum Accepted Temperature :    -273.00
Maximum Accepted Temperature :    3000.00

FAHTS
Step      Time      Accum.      Max heat      Max      Min
no.        (s)      energy      input      temp.      temp.
              (J)      (W/m2)      (degr C)      (degr C)
1          36.00    8.250E+04    1.13E+03     16.0       16.0
2          72.00    3.780E+05    2.62E+03     19.5       19.4
3         108.00    9.549E+05    4.68E+03     26.2       26.2
4         144.00    1.832E+06    6.41E+03     36.5       36.5
5         180.00    2.980E+06    8.10E+03     50.0       50.0
6         216.00    4.273E+06    8.25E+03     65.2       65.1
7         252.00    5.566E+06    8.09E+03     80.4       80.3
8         288.00    6.855E+06    8.22E+03     95.5       95.4
9         324.00    8.142E+06    8.05E+03    110.7      110.5
10        360.00    9.425E+06    8.18E+03    125.7      125.6
-
90       3240.00    5.164E+07    9.15E+02     623.8      618.0
91       3276.00    5.168E+07    9.00E+02     624.3      618.5
92       3312.00    5.173E+07    8.85E+02     624.8      619.0
93       3348.00    5.177E+07    8.71E+02     625.3      619.5
94       3384.00    5.181E+07    8.58E+02     625.8      619.9
95       3420.00    5.184E+07    8.46E+02     626.2      620.4
96       3456.00    5.188E+07    8.35E+02     626.6      620.8
97       3492.00    5.191E+07    8.24E+02     627.0      621.1
98       3528.00    5.194E+07    8.13E+02     627.3      621.5
99       3564.00    5.197E+07    8.04E+02     627.7      621.8
100      3600.00    5.199E+07    7.94E+02     628.0      622.1

```

Figure 2.1-4: Extract of FAHTS out file, *resf.out*.

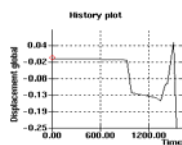


Figure 2.1-5: The global displacement history.

2.2 Single beam exposed to fire on three sides.

Key words:

Ex. 2: Alum Box Partly exp. Fully prot. Fire exp. b) Protection

Material: Aluminium AA 6082-T6. Property reduction due to increased temperature, according to Eurocode 1999 Part 1-2.

Structure: Single beam, aluminium, mid span force, 3-side exposure, full protection.

Similar beam as in the previous example, but different fire exposure.

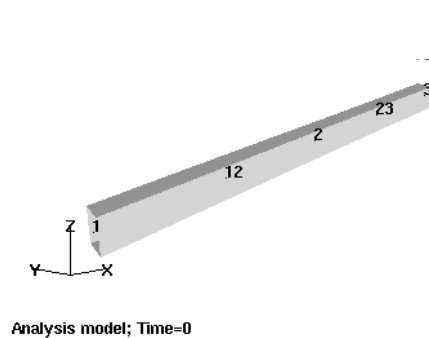


Figure 2.2-1: Model of the structure, with node- and element numbers.

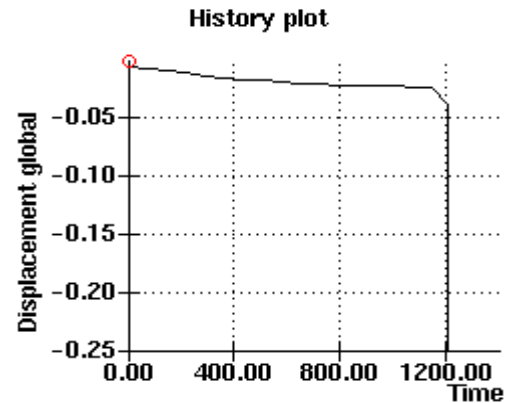


Figure 2.2-2: The global displacement history.

Problem:

Beam partly exposed to fire; three sides. The top is not exposed. Full protection. Different from default values.

Solution:

EXP_BOX contains information about which sides of the profiles that are exposed to fire and which sides that are protected by use of insulation.

0 = not exposed, or not protected.

1 = exposed at the "outside" of the profile, or protected at the "outside" of the profile.

2 = exposed at the "inside" of the profile, or protected at the "inside".

3 = (1+2) exposed both "inside" and "outside", or protected both "inside" and "outside".

Because the profile in this example is a box, it will not be exposed or protected on the "inside". If the box-profile is filled with something different than air, this will be taken account for in the INSIDPAR command. See Basic commands.

HEAD	F A H T S temperature simulations							
	Single beam, box cross-section							
	+							

	TYPE OF FIRE AND EXPOSURE							

HCFIRE	! Hydrocarbon fire.							
	* Exposure *				* Passive fire protection *			
	top	L	R	bott	top	L	R	bott
EXP BOX	0	1	1	1	1	1	1	1

Figure 2.2-3: Control file for FAHTS, *fahts.fem*.

```

*****
***** F A H T S *****
*****
Version 3.1 / Release 98-03-09

FAHTS
Step      Time      Accum.      Max heat      Max      Min
no.        (s)      energy      input      temp.      temp.
              (J)              (W/m2)      (degr C)      (degr C)
1          36.00    6.338E+04    1.13E+03     16.0       14.9
2          72.00    2.972E+05    2.62E+03     19.4       15.3
3         108.00    7.562E+05    4.69E+03     26.2       16.5
-
98        3528.00    4.609E+07    2.69E+03     578.7      513.5
99        3564.00    4.614E+07    2.68E+03     579.2      514.2
100       3600.00    4.619E+07    2.66E+03     579.7      514.9

===== F A H T S   A N A L Y S I S   C O M P L E T E D =====

```

Figure 2.2-4: Extract of FAHTS out file, *resf.out*.

2.3 Single beam exposed to fire on one side.

Key words:

Ex. 3: Alum Box Partly exp. Fully prot. Fire exp. c) Protection

Material: Aluminium AA 6082-T6. Property reduction due to increased temperature, according to Eurocode 1999 Part 1-2.

Structure: Single beam, aluminium, mid span force, 1-side exposure, full protection.

Similar beam as in the two previous examples, but different fire exposure.

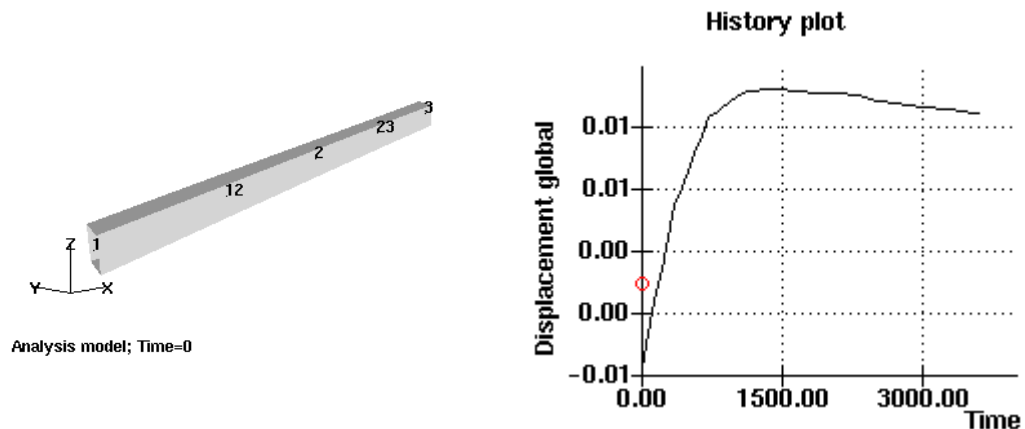


Figure 2.3-1: Structural model, with node- and element numbers.

Figure 2.3-2: The global displacement history.

HEAD	F A H T S temperature simulations							
	Single beam, box cross-section							
	+							

	TYPE OF FIRE AND EXPOSURE							

HCFIRE	! Hydrocarbon fire.							
	* Exposure *				* Passive fire protection *			
	top	L	R	bott	top	L	R	bott
EXP BOX	1	0	0	0	1	1	1	1

Figure 2.3-3: Control file for FAHTS, *fahts.fem*.

***** F A H T S *****					

FAHTS	Time	Accum.	Max heat	Max	Min
Step	(s)	energy	input	temp.	temp.
no.		(J)	(W/m2)	(degr C)	(degr C)
1	36.00	5.804E+03	1.13E+03	15.9	14.8
2	72.00	5.432E+04	2.63E+03	18.9	14.6
-					
99	3564.00	1.769E+07	7.79E+03	286.5	180.7
100	3600.00	1.778E+07	7.77E+03	287.5	181.7
===== F A H T S A N A L Y S I S C O M P L E T E D =====					

Figure 2.3-4: Extract of FAHTS out file, *resf.out*.

2.4 Single beam exposed to fire and protected on three sides.

Key words:

Ex. 4: Alum Box Partly exp. Partly prot. Fire exp. d) Protection

Material: Aluminium AA 6082-T6. Reduction of material properties due to increased temperature, according to Eurocode 1999 Part 1-2.

Structure: Single beam, aluminium, mid span force, 3-side exposure, 3-side insulation.

Similar beam as in the three previous examples, but different fire exposure and insulation.

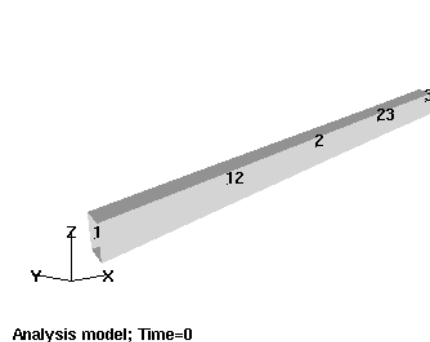


Figure 2.4-1: Model of the beam, with node- and element numbers.

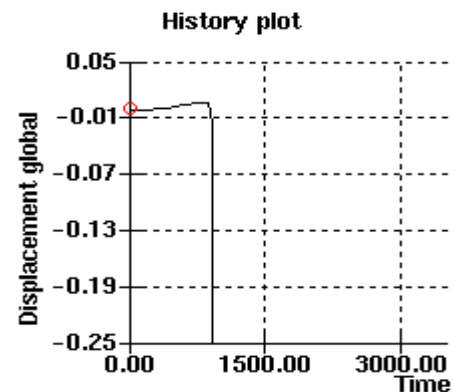


Figure 2.4-2: The global displacement history.

HEAD	F A H T S temperature simulations							
	Single beam, box cross-section							

	TYPE OF FIRE AND EXPOSURE							

HCFIRE	! Hydrocarbon fire.							
	* Exposure *				* Passive fire protection *			
	top	L	R	bott	top	L	R	bott
EXP_BOX	0	1	1	1	0	1	1	1

Figure 2.4-3: Control file for FAHTS, *fahts.fem*.

***** F A H T S *****					

FAHTS	Time	Accum.	Max heat	Max	Min
Step		energy	input	temp.	temp.
no.	(s)	(J)	(W/m2)	(degr C)	(degr C)
1	36.00	6.400E+04	1.13E+03	16.0	15.0
2	72.00	2.985E+05	2.62E+03	19.4	15.4
99	3564.00	4.462E+07	3.14E+03	562.2	493.3
100	3600.00	4.465E+07	3.13E+03	562.5	493.6
=====	F A H T S	A N A L Y S I S	C O M P L E T E D	=====	

Figure 2.4-4: Extract of FAHTS out file, *resf.out*.

2.5 Single beam exposed to fire on three sides.

Key words:

Ex. 5: Alum I-prof Partly exp. Partly prot. Fire exp. e) Protection

Material: Aluminium AA 6082-T6. Reduction of material properties due to increased temperature, according to Eurocode 1999 Part 1-2.

Structure: Single beam, aluminium, I-profile, mid span force, 3-side exposure, 3-side insulation.

Similar beam as in the four previous examples, except from different profile, fire exposure and insulation.

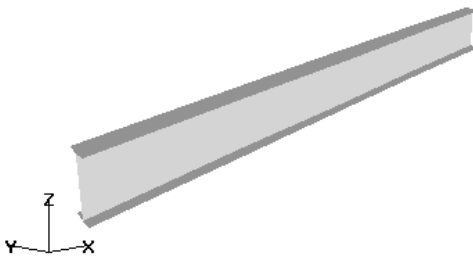


Figure 2.5-1: Model of the beam, with node- and element numbers.

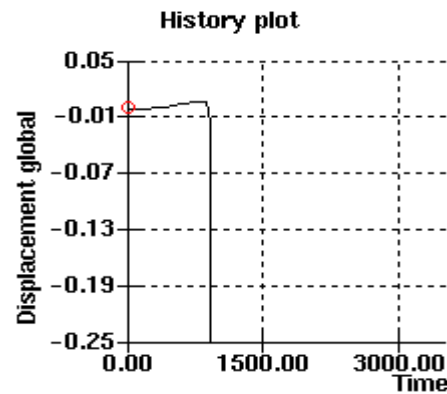


Figure 2.5-2: The global displacement history.

Problem:

Beam with I-profile. Partly exposed and partly protected.

Solution:

Since this is an I-profile it has an "inside" and an "outside". EXP_IPRO tells that the top of the profile is exposed and protected on the "inside", but not on the "outside". The web and the bottom are exposed and protected on both the "inside" and the "outside".

For the web of an I-profile the numbers 2 and 3 means the same; protected, or exposed on both sides. 1 is not an option for the web, because it has no "outside".

```

'-----
'                                TYPE OF FIRE AND EXPOSURE
'-----
HCFIRE      ! Hydrocarbon fire.
'
'      * Exposure *
'      top      web      bott      * Passive fire protection *
'      2        3        3        top      web      bott
EXP_IPRO    2        3        3        2        3        3
'
'      * Initial temperature in the fire compartment *
'      initemp
INITEMP     15.0
'
'      * Exposure time *
'      end-time (s)      nstep      resinc (s)
TEMPSIM     3600        100        360

```

Figure 2.5-3: Control file for FAHTS, *fahts.fem*.

```

*****
***** F A H T S *****
*****
Version 3.1 / Release 98-03-09
Licenced to : SINTEF

===== A N A L Y S I S   P A R A M E T E R S   =====

FAHTS
Step      Time      Accum.      Max heat      Max      Min
no.        (s)      energy      input      temp.      temp.
              (J)              (W/m2)      (degr C)  (degr C)
1         36.00    1.235E+05    2.34E+03     16.5      15.2
2         72.00    5.497E+05    5.31E+03     21.6      16.5
3        108.00    1.404E+06    9.43E+03     30.8      20.8
4        144.00    2.775E+06    1.29E+04     44.3      30.2
5        180.00    4.720E+06    1.88E+04     61.0      47.3
6        216.00    7.012E+06    2.01E+04     79.3      69.0
7        252.00    9.338E+06    2.14E+04     97.5      91.4
8        288.00    1.170E+07    2.27E+04    118.4     110.8
9        324.00    1.411E+07    2.37E+04    141.3     129.2
10       360.00    1.655E+07    2.44E+04    164.5     147.7
-
91       3276.00    1.065E+08    3.20E+03     967.8     892.2
92       3312.00    1.066E+08    3.17E+03     970.4     892.6
93       3348.00    1.068E+08    3.13E+03     973.0     893.0
94       3384.00    1.070E+08    3.10E+03     975.5     893.5
95       3420.00    1.072E+08    3.07E+03     977.9     893.9
96       3456.00    1.073E+08    3.05E+03     980.2     894.3
97       3492.00    1.075E+08    3.02E+03     982.4     894.6
98       3528.00    1.076E+08    2.99E+03     984.6     895.0
99       3564.00    1.078E+08    2.96E+03     986.7     895.3
100      3600.00    1.079E+08    2.94E+03     988.8     895.7

===== F A H T S   A N A L Y S I S   C O M P L E T E D   =====

```

Figure 2.5-4: Extract of FAHTS out file, *resf.out*.

2.6 Beam with displacement limitation and temp. dependant material properties.

Ex. 6: Alum I-prof Partly exp. Partly prot. Displ. lim. Temp. dep.

Material: Aluminium AA 6082-T6. Reduction of material properties due to increased temperature, according to Eurocode 1999 Part 1-2.
Structure: Single beam, aluminium, I-profile, mid span force, 3-side exposure, 3-side insulation.

Same beam as in the previous example.

This example shows the use of displacement limitation and temperature dependent material properties given by the user.

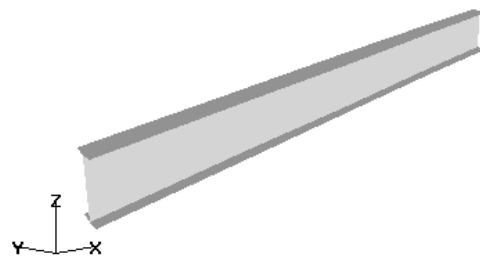


Figure 2.6-1: Model of the beam, with node- and element numbers.

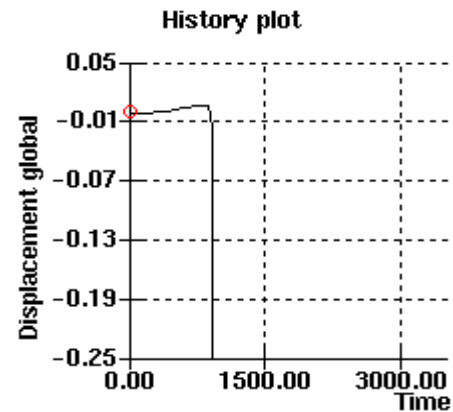


Figure 2.6-2: The global displacement's development over time.

Problem:

Partly exposed beam, with partly protection. Find time and temperature for a max displacement of 250 mm. The temperature dependent reduction of the material properties, is user defined.

Solution:

The condition for max displacement 250 mm, is given in the factor for *mxdisp* in CICYFOS. The max displacement is given in meters in the Z-direction for this specific beam.

The temperature dependency defined by the user is given in USERTDEP and TEMPDEPY. USERTDEP specifies the identity of the material the reduction apply to, and the identity of the reduction curves for each material property. In TEMPDEPY the reduction curves are given as co-ordinates. The reduction in this example is given as a factor. This factor is multiplied with the material property given in MISOIPE (in USFOS Control file).

```

HEAD          Fire capacity of single aluminium beam
              USFOS analysis
              I-profile
,
DETEROFF
CMAXSTEP      1200
XFOSFULL
-----
LOAD HISTORY AND DEFORMATION CONTROL
-----
CICYFOS      nloads  npostp  mxpstp  mxpdis
              10      100      1        1
,
              lcase lfact mxld  mxdisp  nstep  minstp
              2    1.0  1.0    0.0     10    0.001  ! Gravity.
              1    1.0  2.0   -0.250   100    0.001  ! Nodeload.
              4    0.02 1.0   -0.250   200    0.001  ! Fahts Fire Load
              5    0.02 1.0   -0.250   200    0.001  ! Fahts Fire Load
              6    0.02 1.0   -0.250   200    0.001  ! Fahts Fire Load
              7    0.02 1.0   -0.250   200    0.001  ! Fahts Fire Load
,
ncnodes
CNODES       1
,
              nodex idof dfact
              2     3     1
-----
MATERIAL PROPERTIES
-----
matno  E-mod  poiss  yield  density  Therm. exp
MISOIEP  1  70000.0E6  0.3  255.0E6  2700.0  2.3E-5
,
* Temperature dependent property reduction *
matno  Dep E  Dep Yield
USERTDEP  1    1    2
,
curve_id  Temp  Red  ! Dependency of E-mod
TEMPDEPY  1    20  1.0
           50  0.99
           100 0.97
           150 0.93
           200 0.86
           250 0.78
           300 0.68
           350 0.54
           400 0.40
           550 0.0
,
curve no  Temp  Red  ! Dependency of Yield
TEMPDEPY  2    20  1.0
           100  1.0
           150 0.79
           200 0.65
           250 0.38
           300 0.20
           350 0.11
           550 0.0

```

Figure 2.6-3: USFOS Control file, *usfos.fem*.

2.7 Cantilever beam fully exposed to fire.

Key words:

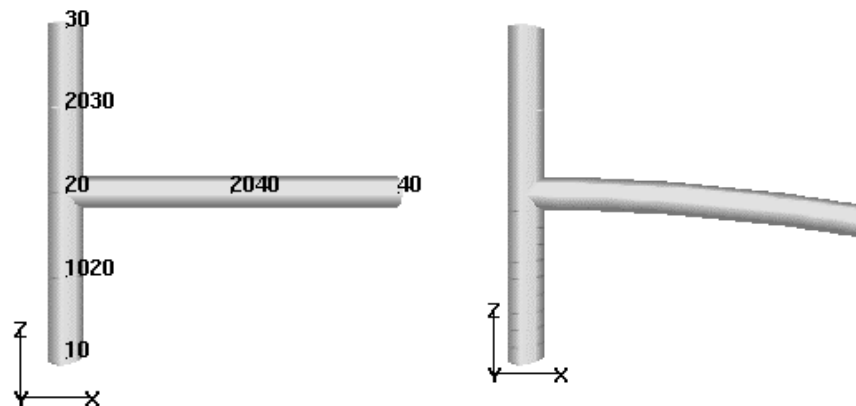
Ex. 7: Steel Pipe Partly exp. Unprotected User source Radiation

Material: Steel.

Structure: Cantilever beam, nodal force in node 40, full exposure, not insulated.

This example shows the use of user defined radiation fields.

The structure is modelled with three elements as shown in Figure 2.7-1.



Analysis model; Time=450

Figure 2.7-1: Structural model with node and element numbers.

Figure 2.7-2: Deformed model.

Column length	:	10.000 m
Diameter (outer)	:	1.100 m
Thickness	:	0.020 m

Beam length	:	10.000 m
Diameter (outer)	:	1.000 m
Thickness	:	0.020 m

Yield stress	:	$\sigma = 248 \text{ MPa}$
Youngs modulus	:	$E = 210000 \text{ MPa}$

Nodal force, node 40	:	10 kN
Fire load	:	User defined radiation field

Problem:

Structure with perpendicular radiation on the cantilever beam.
The structure is exposed to radiation intensity of 10000 W/m^2 , which is constant.

Solution:

USERFIRE describes the problem by giving:

- constant exposure; history = 0,
- where the source is located; with X-coord. = 5 m, Y-coord. = 0 m and Z-coord. = 10000 m,
- intensity at radius distance from the source; 10000 W/m^2 , and
- reference radius from the source used to define the intensity; 10000 m.

By setting a big distance, the rays, when they reach the structure, are almost parallel, and thereby perpendicular to the beam.

The distance from the source to the structure is equal to the radius distance for the intensity. Thereby, the intensity at the structure is 10000 W/m^2 .

Because the thermal elongation on top of the beam is bigger than under, the end of the beam will bent down. When the underside gets hotter and the temperature increment through the cross-section is reduced, the beam will bend up again and straighten out a bit.

```

HEAD
      F A H T S  temperature simulation
                  Beam + Column
'
MOVIEPRI      ! Override default mesh refinement.
SHAPFACT      ! Shape factor calc. is switched on.
'
      n_length  n_circ
TEMPMESH      2      16
'
-----
      USER DEFINED FIRE
-----
'
      history    x      y      z      ref_intensity  ref_radius
USERFIRE        0      5      0.0  10.0E3      10000.0      10.0E3
'
-----
      ANALYSIS TIME AND TEMPERATURE STEPS
-----
'
      end-time (s)  nstep      resinc (s)
TEMPSIM           10800.0      300      900.0
'
-----
      THE MATERIAL'S THERMAL PARAMETERS
-----
'
      id      rho(kg/m3)  c (J/kgK)  k (W/mK)  emiss
THERMPAR       10000      7850.0      500.0      50.0      0.8

```

Figure 2.7-3: FAHTS Control file, *fahts.fem*.

```

HEAD          FAHTS example : Column and cantilever beam
              U S F O S
              SINTEF div of Structural Engineering
'-----
'             AMOUNT OF INPUT AND OUTPUT PRINTED TO FILE
'-----
'             inpri   outpri   termpri
CPRINT        1       1       1
'-----
'             LOAD HISTORY AND DEFORMATION CONTROL
'-----
'             nloads   npostp   mxpstp   mxpdis
CUSFOS        30      15       1.00    0.05
'             lcomb    lfact    mxld     nstep    minstp
'             4        0.5      1.0      15       0.05
'             5        0.5      1.0      15       0.05
'             6        0.5      1.0      15       0.05
'             7        0.5      1.0      15       0.05
'             8        0.5      1.0      15       0.05
'             9        0.5      1.0      15       0.05
'             10       0.5      1.0      15       0.05
'             11       0.5      1.0      15       0.05
'             12       0.5      1.0      15       0.05
'             13       0.5      1.0      15       0.05
'
'             ncnods
CNODES        1
'             nodex    idof     dfact
'             40      3        1.
'-----
'             INELASTIC MATERIAL PROPERTIES
'-----
'             matno    E-mod    poiss    yield    density    therm. exp.
MISOIEP      10000    210000.0E6    0.3    248.E6    7850.0    1.4E-05

```

Figure 2.7-4: USFOS Control file, *usfos.fem*.

```

*****
***** F A H T S *****
*****

Version 3.1 / Release 98-03-09
S I N T E F

F A H T S temperature simulation
Beam + Column

Number of nodal points      = 17
Number of elements         = 16
Minimum Accepted Temperature : -273.00
Maximum Accepted Temperature : 3000.00

FAHTS
Step no.      Time      Accum.      Max heat      Max      Min
              (s)       energy       input         temp.      temp.
                      (J)       (W/m2)        (degr C)   (degr C)
1             36.00    2.864E+06    7.85E+03      23.7      19.9
2             72.00    5.724E+06    7.82E+03      27.4      19.9
3            108.00    8.578E+06    7.80E+03      31.0      19.8
4            144.00    1.143E+07    7.77E+03      34.6      19.8
5            180.00    1.427E+07    7.75E+03      38.3      19.8
6            216.00    1.710E+07    7.72E+03      41.8      19.7
7            252.00    1.993E+07    7.70E+03      45.4      19.7
8            288.00    2.275E+07    7.67E+03      48.9      19.7
9            324.00    2.556E+07    7.64E+03      52.5      19.6
10           360.00    2.837E+07    7.61E+03      56.0      19.6
-
-
-
290          10440.00  4.057E+08    8.24E+02      323.6     18.9
291          10476.00  4.061E+08    8.22E+02      323.6     18.9
292          10512.00  4.066E+08    8.19E+02      323.7     18.9
293          10548.00  4.070E+08    8.17E+02      323.7     18.9
294          10584.00  4.074E+08    8.15E+02      323.8     18.8
295          10620.00  4.078E+08    8.13E+02      323.8     18.8
296          10656.00  4.082E+08    8.11E+02      323.9     18.8
297          10692.00  4.086E+08    8.09E+02      323.9     18.8
298          10728.00  4.091E+08    8.06E+02      323.9     18.8
299          10764.00  4.095E+08    8.04E+02      324.0     18.8
300          10800.00  4.099E+08    8.02E+02      324.0     18.8
===== F A H T S   A N A L Y S I S   C O M P L E T E D =====

```

Figure 2.7-5: Extract of FAHTS out file, *resf.out*.

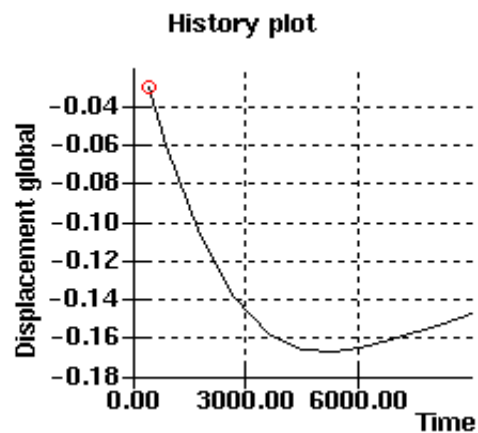


Figure 2.7-6: The global displacement's development over time.

2.8 Cantilever beam exposed to two radiation sources.

Ex. 8: Steel Pipe Partly exp. Unprotected User source Radiation

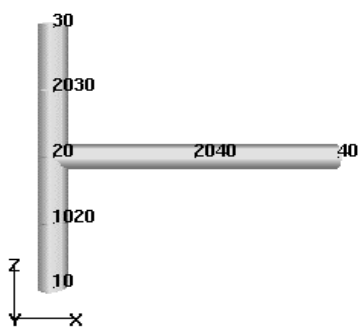
Material: Steel.

Structure: Cantilever beam, nodal force in node 40, full exposure, not insulated.

Similar structure as in the previous example.

This example shows the use of user defined radiation fields.

The structure is modelled with three elements (with refinement) as shown in Figure 2.8-1.



Analysis model; Time=450

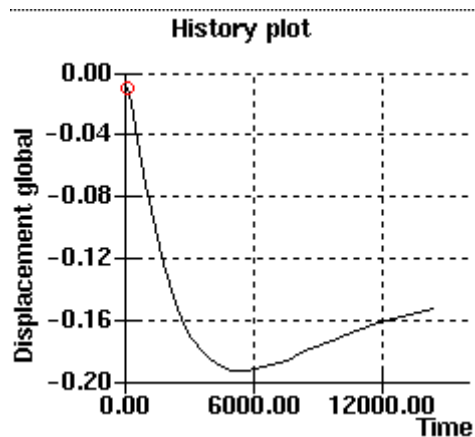


Figure 2.8-1: Model of the structure, with node- and element numbers.

Figure 2.8-2: The global displacement history.

Column length	:	10.000 m
Diameter (outer)	:	1.100 m
Thickness	:	0.020 m
Beam length	:	10.000 m
Diameter (outer)	:	1.000 m
Thickness	:	0.020 m
Yield stress	:	$\sigma = 248 \text{ MPa}$
Youngs modulus	:	$E = 210000 \text{ MPa}$
Nodal force, node 40	:	10 kN
Fire load	:	User defined radiation field

Problem:

Two radiation sources, one is constant, the other is time dependent.

Solution:

Defines two sources with different time histories. The constant heat source is defined by giving it a *hist_id* 0. This tells that the source is constant. The reference intensity of the USERFIRE with *hist_id* 100 is set to 1.0. This is because the intensity is given in the TIMEHIST command. The *ref_intensity* factor is multiplied with the *factor* from TIMEHIST and the result is the total intensity at the given time.

```

HEAD
      F A H T S  temperature simulation
      Beam  + Column
-----
      USER DEFINED FIRE
-----
      hist_id  x      y      z      ref_intensity  ref_radius
USERFIRE      0      5      0.0  10.0E3    10000.0      10.0E3
USERFIRE     100      5      0.0  10.0E3      1.0      10.0E3
-----
      hist_id  Type  time  factor
TIMEHIST     100      1      0.0      0.0
                        300.0      0.0
                        720.0     800.0
                        1020.0    1700.0
                        1200.0    1550.0
                        1500.0    1400.0
                        1980.0    1300.0
                        2100.0    1000.0
                        2700.0     250.0
                        3600.0     150.0
                        7200.0     100.0
                        10800.0      0.0
                        14400.0      0.0

```

Figure 2.8-3: FAHTS Control file, *fahts.fem*.

```

*****
***** F A H T S *****
*****
Version 3.1 / Release 98-03-09
S I N T E F

FAHTS
Step   Time   Accum.   Max heat   Max   Min
no.    (s)     energy   input      temp.  temp.
              (J)      (W/m2)   (degr C) (degr C)
1     144.00  1.145E+07  7.85E+03   34.7   19.8
2     288.00  2.286E+07  7.75E+03   49.2   19.6
3     432.00  3.429E+07  7.83E+03   63.5   19.6
4     576.00  4.591E+07  7.92E+03   77.8   19.6
5     720.00  5.771E+07  8.00E+03   92.1   19.7
6     864.00  6.976E+07  8.18E+03  106.5   19.9
7    1008.00  8.215E+07  8.33E+03  121.1   19.9
8    1152.00  9.455E+07  8.07E+03  135.5   19.8
9    1296.00  1.066E+08  7.78E+03  149.2   19.8
10   1440.00  1.184E+08  7.48E+03  162.2   19.8
-
91   13104.00  4.347E+08  6.94E+02   326.6   18.7
92   13248.00  4.356E+08  6.92E+02   326.7   18.7
93   13392.00  4.365E+08  6.89E+02   326.8   18.6
94   13536.00  4.374E+08  6.87E+02   326.8   18.6
95   13680.00  4.382E+08  6.84E+02   326.9   18.6
96   13824.00  4.390E+08  6.82E+02   327.0   18.6
97   13968.00  4.398E+08  6.80E+02   327.0   18.6
98   14112.00  4.406E+08  6.78E+02   327.1   18.6
99   14256.00  4.413E+08  6.76E+02   327.2   18.6
100  14400.00  4.420E+08  6.74E+02   327.2   18.6

===== F A H T S   A N A L Y S I S   C O M P L E T E D =====

```

Figure 2.8-4: Extract of FAHTS out file, *resf.fem*.

2.9 Simple portal frame.

Key words:

Ex. 9: Steel I-prof Fully exp. Unprotected User source Temp. hist.

Material: Steel.

Structure: Simple frame, nodal forces in nodes 11 and 12, distributed load on the beam, temperature rise in the environment. Not protected.

This example shows the use of user defined constant temperature history.

The structure is modelled with elements as shown in Figure 2.9-1.

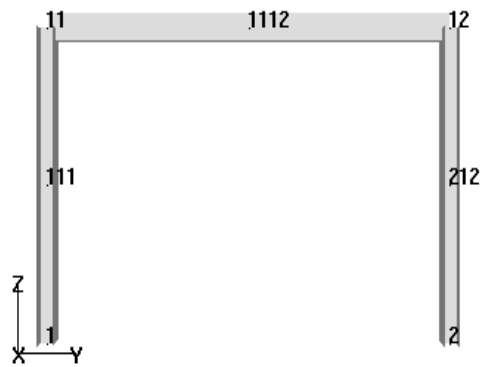


Figure 2.9-1: Structural model.

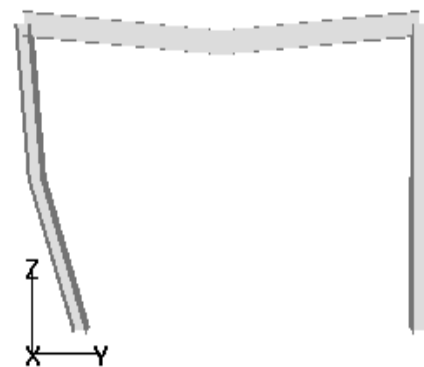


Figure 2.9-2: Deformed model.

Column	length :	3.500 m
	ight :	0.206 m
	width :	0.204 m
Beam	length :	5.500 m
	height :	0.304 m
	width :	0.165 m
Yield stress :		$\sigma = 275 \text{ MPa}$
Youngs modulus :		$E = 205000 \text{ MPa}$
Nodal forces:		
	node 11:	500 kN
	node 12:	500 kN
Distributed force		
	beam :	25.4 kN/m
Heat source :		User defined temperature history

Problem:

Constant temperature in the environment around the structure.

Solution:

USERTEMP describes the number of the temperature/time history to be used, *hist_id* = 0 (constant time history). The *field_temp* is set to 600 Celsius degrees, which is the temperature in the environment.

```

HEAD          F A H T S  temperature simulation

                Simple Portal Frame
MOVIEPRI
-----
'              TYPE OF FIRE AND EXPOSURE
'-----
'              initemp
'              20.0
'
'              hist_id    field_temp
'              0          600
'
'-----
'              ANALYSIS TIME AND TEMPERATURE STEPS
'-----
'              end-time (s)    nstep    resinc (s)
'              7200.0         50       120.0
'
'-----
'              THE MATERIAL'S THERMAL PARAMETERS
'-----
'              mat_id  rho(kg/m3)  c (J/kgK)  k (W/mK)  emiss  convection
'              1       7850.0      450.0      54.0      0.8    20
'
'              mat_id  rho(kg/m3)  c (J/kgK)  k (W/mK)  dummy  dummy
'              1       1.273       0.7        0.0      0.0    0.0
'              INSIDPAR

```

Figure 2.9-3: FAHTS Control file, *fahts.fem*.

```

HEAD          Portal Frame
                U S F O S  progressive collapse analysis
                NUS/ SINTEF Structural Engineering
TMPLOCON      1
BANANA
STEELTDEP     ! Temperature dependent material properties, reduction.
'             rest.  xfos  out-file
'             0     -10   -50
'-----
'             LOAD HISTORY AND DEFORMATION CONTROL
'-----
'             nloads  npostp  mxpstp  mxpdis
'             10      100     1.00    0.05
'             lcomb   lfact   mxld    nstep   minstp
'             1       0.5     1.0     10      0.01 ! Dead weight
'             4       0.05    1.0     100     0.01 ! Fire load
'             5       0.05    1.0     100     0.01 ! Fire load
'             6       0.02    1.0     100     0.01 ! Fire load
'             7       0.01    1.0     100     0.01 ! Fire load
'             8       0.01    1.0     100     0.01 ! Fire load
'             9       0.01    1.0     100     0.01 ! Fire load
'             10      0.01    1.0     100     0.01 ! Fire load
'             -
'             23      0.01    1.0     100     0.01 ! Fire load
'
'             ncnods
'             1
'             CNODES
'             nodex   idof    dfact
'             11      1       1.
'             11      2       1.
'             11      3       1.
'-----
'             INELASTIC MATERIAL PROPERTIES
'-----
'             matno   E-mod    poiss   yield    density  therm. exp.
'             1       205000.0E6  0.3    275.0E6   7850     1.4E-05
'             MISOIEP

```

Figure 2.9-4: USFOS Control file, *usfos.fem*.

***** F A H T S *****					
Version 3.1 / Release 98-03-09					
S I N T E F					
FAHTS Step no.	Time (s)	Accum. energy (J)	Max heat input (W/m2)	Max temp. (degr C)	Min temp. (degr C)
1	144.00	8.437E+07	7.53E+04	434.7	257.2
2	288.00	1.537E+08	5.59E+04	709.6	513.5
3	432.00	1.777E+08	1.67E+04	750.8	667.0
4	576.00	1.583E+08	-2.14E+04	664.3	526.9
5	720.00	1.406E+08	1.85E+04	595.2	472.2
6	864.00	1.473E+08	2.80E+04	604.7	560.3
7	1008.00	1.590E+08	1.06E+04	667.1	582.6
8	1152.00	1.585E+08	3.52E+03	633.7	609.1
9	1296.00	1.520E+08	-2.60E+03	613.1	550.0
10	1440.00	1.517E+08	1.31E+04	601.0	564.2
-					
41	5904.00	1.545E+08	4.38E+01	600.0	599.9
42	6048.00	1.545E+08	2.69E+01	600.1	600.0
43	6192.00	1.545E+08	4.69E-02	600.1	600.0
44	6336.00	1.545E+08	-1.63E-02	600.0	600.0
45	6480.00	1.545E+08	1.19E+01	600.0	599.9
46	6624.00	1.545E+08	1.77E+01	600.0	600.0
47	6768.00	1.545E+08	8.95E-02	600.0	600.0
48	6912.00	1.545E+08	9.22E-03	600.0	600.0
49	7056.00	1.545E+08	-2.65E-03	600.0	600.0
50	7200.00	1.545E+08	8.95E+00	600.0	600.0
===== F A H T S A N A L Y S I S C O M P L E T E D =====					

Figure 2.9-5: Extract of result file from FAHTS analysis, *resf.out*.

2.10 Simple portal frame.

Key words:

Ex. 10: Steel I-prof Fully exp. Unprotected User source Temp. hist.

Material: Steel.

Structure: Simple frame, nodal forces in nodes 11 and 12, distributed load on the beam, temperature rise in the environment. Not protected.

This example shows the use of user defined temperature history.

The structure is modelled with elements as shown in Figure 2.10-1.

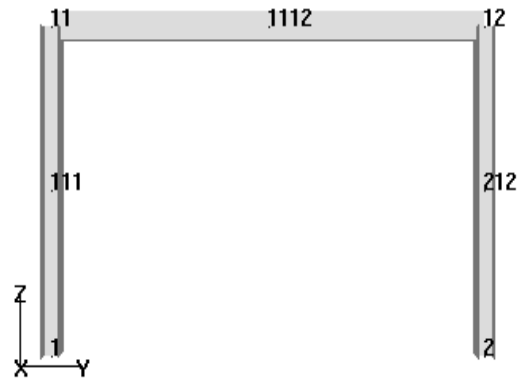


Figure 2.10-1: Structural model.

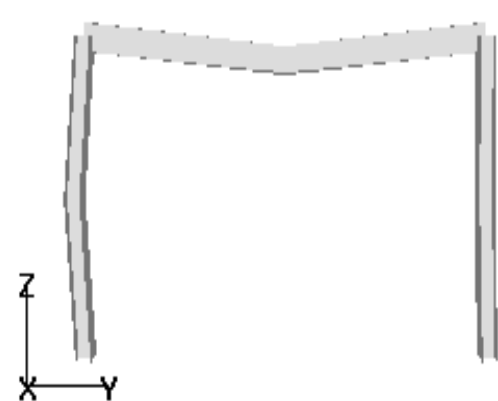


Figure 2.10-2: Deformed model.

Column	length :	3.500 m
	height :	0.206 m
	width :	0.204 m
Beam	length :	5.500 m
	height :	0.304 m
	width :	0.165 m
Yield stress :		$\sigma = 275 \text{ MPa}$
Youngs modulus :		$E = 205000 \text{ MPa}$
Nodal forces:		
	node 11:	500 kN
	node 12:	500 kN
Distributed force:		
	beam :	25.4 kN/m
Heat source :	User defined temperature history	

Problem:

Increasing temperature in the environment around the structure.

Solution:

USERTEMP describes the number of the temperature/time history to be used, **hist_id** = 100, and a factor, **field_temp**, to be multiplied with the **factor** from TIMEHIST to give the temperature. The **field_temp** factor is set to 1.0 in this example because the **factor** in TIMEHIST describes the temperature in Celsius degrees. This is the recommended way to do it, because of the readability.

TIMEHIST sets the variation of the temperature over time for the specific history. The factor **Type** defines the type of points on the curve that are used. The number 1 defines discrete points.

USERTEMP can be extended if the heat source only affects a part of the structure. See the following example.

HEAD	F A H T S temperature simulation					
	Simple Portal Frame					
MOVIEPRI						

	TYPE OF FIRE AND EXPOSURE					

	initemp					
INITEMP	20.0					
	hist_id	field_temp				
USERTEMP	100	1.0				
	hist_id	Type	time	factor		
TIMEHIST	100	1	0.0	20.0		
			1500.0	697.0		
			2160.0	734.0		
			2940.0	697.0		
			4560.0	514.0		
			12960.0	20.0		
			100000.0	20.0		

	ANALYSIS TIME AND TEMPERATURE STEPS					

	end-time (s)	nstep	resinc (s)			
TEMPSIM	14400.0	200	360.0			

	THE MATERIAL'S THERMAL PARAMETERS					

	mat_id	rho (kg/m3)	c (J/kgK)	k (W/mK)	emiss	convection
THERMPAR	1	7850.0	450.0	54.0	0.8	20
	mat_id	rho (kg/m3)	c (J/kgK)	k (W/mK)	dummy	dummy
INSIDPAR	1	1.273	0.7	0.0	0.0	0.0

Figure 2.10-3: FAHTS Control file, *fahts.fem*.

```

HEAD          Portal Frame
              U S F O S progressive collapse analysis
              NUS/ SINTEF Structural Engineering
TMPLOCON      1
BANANA
STEELTDEP     ! Temperature dependent material properties, reduction.
rest. xfos out-file
CSAVE         0      -10      -50
-----
LOAD HISTORY AND DEFORMATIONCONTROL
-----
CUSFOS        nloads  npostp  mxpstp  mxpdis
              10      100      1.00    0.05
              lcomb   lfact   mxld    nstep   minstp
              1       0.5    1.0     10      0.01 ! Dead weight
              4       0.05   1.0     100     0.01 ! Fire load
              5       0.05   1.0     100     0.01 ! Fire load
              6       0.02   1.0     100     0.01 ! Fire load
              7       0.01   1.0     100     0.01 ! Fire load
              8       0.01   1.0     100     0.01 ! Fire load
              9       0.01   1.0     100     0.01 ! Fire load
              10      0.01   1.0     100     0.01 ! Fire load
              11      0.01   1.0     100     0.01 ! Fire load
              12      0.01   1.0     100     0.01 ! Fire load
              13      0.01   1.0     100     0.01 ! Fire load
              ncnods
CNODES        1
              nodex   idof    dfact
              11      1       1.
              11      2       1.
              11      3       1.
-----
INELASTIC MATERIAL PROPERTIES
-----
MISOIEP       matno   E-mod    poiss   yield    density  therm. exp.
              1      205000.0E6  0.3     275.0E6  7850     1.4E-05

```

Figure 2.10-4: USFOS Control file, *usfos.fem*.

```

***** F A H T S *****
*****
FAHTS
Step   Time    Accum.    Max heat    Max        Min
no.    (s)       energy    input       temp.      temp.
              (J)       (W/m2)     (degr C)   (degr C)
1      72.00   4.624E+05  1.65E+03    22.3       21.2
2      144.00  1.859E+06  3.35E+03    29.2       25.2
3      216.00  4.190E+06  5.05E+03    40.2       32.1
4      288.00  7.443E+06  6.76E+03    55.3       42.1
5      360.00  1.162E+07  8.52E+03    74.2       55.3
6      432.00  1.673E+07  1.03E+04    96.9       71.8
7      504.00  2.280E+07  1.22E+04    123.6      91.8
8      576.00  2.985E+07  1.42E+04    154.2     115.3
9      648.00  3.789E+07  1.62E+04    188.6     142.4
10     720.00  4.694E+07  1.83E+04    226.8     173.3
-
190    13680.00 2.827E+06 -3.53E+02    33.6       26.2
191    13752.00 2.529E+06 -3.06E+02    32.3       25.4
192    13824.00 2.264E+06 -2.65E+02    31.1       24.6
193    13896.00 2.027E+06 -2.29E+02    30.0       24.0
194    13968.00 1.816E+06 -1.98E+02    29.1       23.5
195    14040.00 1.628E+06 -1.72E+02    28.2       23.0
196    14112.00 1.459E+06 -1.49E+02    27.4       22.6
197    14184.00 1.309E+06 -1.29E+02    26.7       22.3
198    14256.00 1.174E+06 -1.12E+02    26.0       22.0
199    14328.00 1.054E+06 -9.69E+01    25.5       21.7
200    14400.00 9.458E+05 -8.40E+01    24.9       21.5

===== F A H T S   A N A L Y S I S   C O M P L E T E D =====

```

Figure 2.10-5: Extract of result file from FAHTS analysis, *resf.out*.

2.11 3 Bay and 3 Storey frame.

Key words:

Ex. 11: Steel I-prof Partly exp. Unprotected User source Temp. hist.

Material: Steel.

Structure: 3 Bay x 3 Storey frame, nodal forces on top of the columns, distributed load on all horizontal elements, heat source placed on the first floor in the first bay. Not protected.

This example shows the use of user defined temperature history, and the heat source location.

The structure is modelled as shown in Figure 2.11-1.

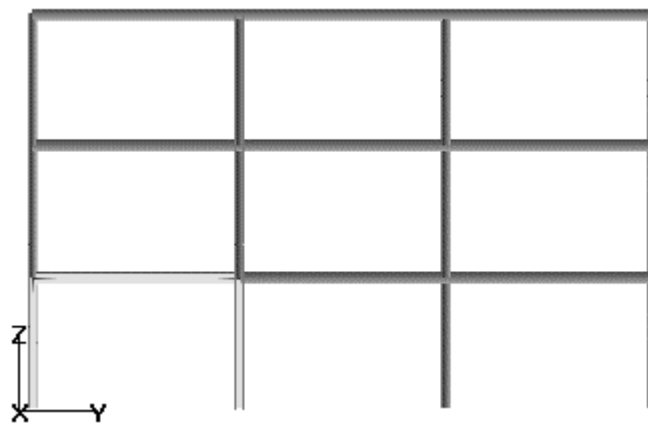


Figure 2.11-1: Structural model.

Column	length :	3.500 m
	height :	0.206 m
	width :	0.204 m
Beam	length :	5.500 m
	height :	0.304 m
	width :	0.165 m

Nodal forces:

end columns	:	75 kN
mid columns	:	151 kN

Distributed forces:

all horizontal elements:	25.4 kN/m
--------------------------	-----------

Heat source	:	User defined temperature history
-------------	---	----------------------------------

Problem:

3 Bay x 3 Storey frame, with heat source on ground level in first bay.

Solution:

This example is similar to the previous, except from a few new conditions in the USERTEMP command.

Gas_Abs is always set to 1.0.

The next conditions define which part of the structure that is exposed to the heat.

There are three ways to do this, and they are described in the FAHTS User's Manual.

The command is called LIMITMOD, but can be integrated in USERTEMP, as in this example.

The **Lim_Type** can be 1, -1, 2, -2, 3 or -3. For more information about the difference here, see FAHTS User's Manual.

In this example the **Lim_Type** is 1. I.e., the part of the structure, which is of interest with respect on temperature distribution, is extracted from the structure with a box defined by two points. When **Lim_Type** is set to 1, all the elements on the inside of the box are included in the transient heat transfer analysis.

The next numbers are the co-ordinates of the two points of the box. To obtain a good readability of the *Control file*, the co-ordinates of the second point are written under the co-ordinates of the first point.

If the heat source is radiation, the command LIMTFIRE is used to extract a part of the structure. See USFOS User's Manual.

```

HEAD          3 Bay x 3 storey frame, Partly Exposed.
              F A H T S temperature simulation
              NUS/SINTEF Structural Engineering

MOVIEPRI
'-----
'              PROFILE MESH
'-----
'              n_length n_web  n_top n_bott
MESHIPRO      8        1      2      2
MESHIPRO      8        6      6      6   1112  111 212
'-----
'              HEAT SOURCE
'-----
'              initemp
INITEMP       20.0
'-----
'              hist_id  field_temp  Gas_Abs  Lim_Type    x      y      z
USERTEMP      100      1.0        1.0        1      -1.0  -1.0  -1.0
              1.0      5.6      3.6
'-----
'              hist_id  Type      time      factor
TIMEHIST      100      1          0.0      20.0
              1500.0    697.0
              2160.0    734.0
              2940.0    697.0
              4560.0    514.0
              12960.0   20.0
              100000.0  20.0
'-----
'              EXPOSURE AND PROTECTION
'-----
'              * Exposure *      * Passive Fire Protection *
'              Geo_Type  top Web bott      top Web bott
EXP_ELEM      2          2  3   3          0   0   0
'              List of elements
'              1112  ! Floor Beam, first floor, first bay
'-----
'              * Exposure *      * Passive Fire Protection *
'              Geo_Type  top Web bott      top Web bott
EXP_ELEM      2          3  3   2          0   0   0
'              List of elements
'              111  ! First bay, column to the left
'-----
'              * Exposure *      * Passive Fire Protection *
'              Geo_Type  top Web bott      top Web bott
EXP_ELEM      2          2  3   3          0   0   0
'              List of elements
'              212  ! First bay, column to the right
'-----
'              ANALYSIS TIME AND TEMPERATURE STEPS
'-----
'              end-time (s)  nstep      resinc (s)
TEMPSIM       7200.0      200        360.0
'-----
'              THE MATERIAL'S THERMAL PARAMETERS
'-----
'              id      rho (kg/m3)  c (J/kgK)  k (W/mK)  emiss  convection
THERMPAR      1      7850.0      450.0      54.0      0.8    20
THERMPAR      2      7850.0      450.0      54.0      0.8    20
'-----
'              id      rho (kg/m3)  c (J/kgK)  k (W/mK)  dummy  dummy
INSIDPAR      1      1.273      0.7        0.0      0      0

```

Figure 2.11-2: FAHTS Control file, *fahts.fem*.

```

*****
***** F A H T S *****
*****

Version 3.1 / Release 98-03-09
S I N T E F

- Analysis initiated at -
98-06-25 09:28:25
Licenced to : SINTEF

FAHTS
Step   Time   Accum.   Max heat   Max   Min
no.    (s)      energy   input      temp.  temp.
              (J)      (W/m2)    (degr C) (degr C)
1      36.00   9.663E+04 8.11E+02   20.7   20.0
2      72.00   3.870E+05 1.64E+03   22.8   19.8
3     108.00   8.698E+05 2.46E+03   26.3   19.6
-
198    7128.00 8.572E+07 1.03E+04   450.2   2.1
199    7164.00 8.520E+07 1.02E+04   447.8   2.1
200    7200.00 8.469E+07 1.01E+04   445.3   2.0

===== F A H T S   A N A L Y S I S   C O M P L E T E D =====

```

Figure 2.11-3: Extract of FAHTS out file, *resf.out*.

2.12 9 Bay and 3 Storey frame.

Ex. 12: Steel I-prof Partly exp. Unprotected User source Temp. hist.

Material: Steel.

Structure: 9 Bay x 3 Storey frame, nodal forces on top of all columns, distributed load on all horizontal elements, heat source placed on the second floor in bay No 5. Partly protected.

This example shows the use of user defined temperature history.

The structure is modelled with nodes as shown in .

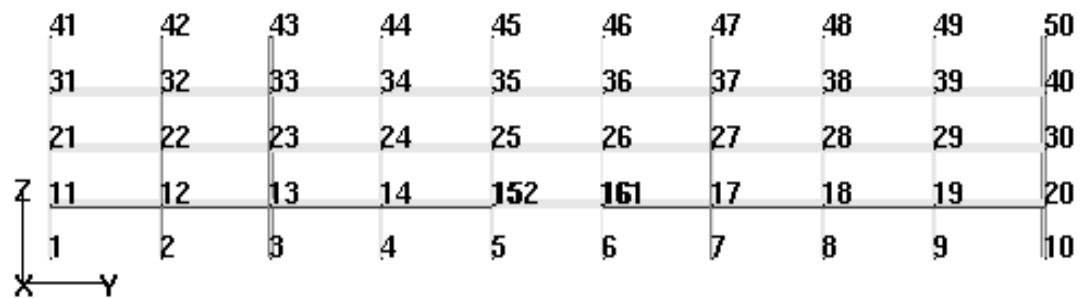


Figure 2.12-1: Structural model.



Figure 2.12-2: Deformed model.

Column	length :	3.600 m
	height :	0.276 m
	width :	0.261 m
Beam	length :	8.000 m
	height :	0.602 m
	width :	0.228 m

Yield stress	:	$\sigma = 275 \text{ MPa}$
Youngs modulus	:	$E = 205000 \text{ MPa}$

Nodal forces:		
node 41 and 50:	:	475.2 kN
node 42-49	:	950.4 kN

Distributed forces:		
on all beams	:	59.4 kN/m

Heat source	:	User defined temperature history
-------------	---	----------------------------------

Problem:

9 Bay x 3 Storey frame. Several different heat sources with temperature/time-histories.

Solution:

Each heat source must be described with a USERTEMP commando line and a time history, TIMEHIST.

Each source gets an identification number, which is equal for the commands, USERTEMP and TIMEHIST.

The sources are defined in the same way as the source in the previous example.

```

HEAD          9 Bay x Four storey frame, Partly Exposed.
              F A H T S temperature simulation
              NUS/SINTEF Structural Engineering
,
MOVIEPRI
GLVIEW
-----
,
              PROFILE MESH
-----
,
              n_length n_web  n_top n_bott
MESHIPRO      8       1       2       2
-----
,
              HEAT SOURCE
-----
,
              initemp
INITEMP       20.0
,
,
              hist field_temp Gas_Abs Lim_Type   x   y   z
USERTEMP      31      1.0      1.0      1      -1.0 15.0 3.5
              1.0      25.0 7.3
,
,
              hist field_temp Gas_Abs Lim_Type   x   y   z
USERTEMP      41      1.0      1.0      1      -1.0 23.0 3.5
              1.0      33.0 7.3
,
,
              hist field_temp Gas_Abs Lim_Type   x   y   z
USERTEMP      51      1.0      1.0      1      -1.0 31.0 3.5
              1.0      41.0 7.3
,
,
              hist field_temp Gas_Abs Lim_Type   x   y   z
USERTEMP      61      1.0      1.0      1      -1.0 39.0 3.5
              1.0      49.0 7.3
,
,
              hist field_temp Gas_Abs Lim_Type   x   y   z
USERTEMP      71      1.0      1.0      1      -1.0 47.0 3.5
              1.0      57.0 7.3
,
,
              id  Type   time   factor
TIMEHIST       31    1     0.0    20.0
              4000.0    20.0
              5500.0    697.0
              6160.0    734.0
              6940.0    697.0
              8560.0    514.0
              16960.0    20.0
              100000.0    20.0
TIMEHIST       41    1     0.0    20.0
              2000.0    20.0
              3500.0    697.0
              4160.0    734.0
              4940.0    697.0
              6560.0    514.0
              14960.0    20.0
              100000.0    20.0
TIMEHIST       51    1     0.0    20.0
              1500.0    697.0
              2160.0    734.0
              2940.0    697.0
              4560.0    514.0
              12960.0    20.0
              100000.0    20.0

```

	id	Type	time	factor			
TIMEHIST	61	1	0.0	20.0			
			2000.0	20.0			
			3500.0	697.0			
			4160.0	734.0			
			4940.0	697.0			
			6560.0	514.0			
			14960.0	20.0			
			100000.0	20.0			
	id	Type	time	factor			
TIMEHIST	71	1	0.0	20.0			
			4000.0	20.0			
			5500.0	697.0			
			6160.0	734.0			
			6940.0	697.0			
			8560.0	514.0			
			16960.0	20.0			
			100000.0	20.0			

ANALYSIS TIME AND TEMPERATURE STEPS							

	end-time (s)	nstep	resinc (s)				
TEMPSIM	7200.0	200	360.0				

THE MATERIAL'S THERMAL PARAMETERS							

	id	rho (kg/m3)	c (J/kgK)	k (W/mK)	emiss	convection	
THERMPAR	1	7850.0	450.0	54.0	0.8	20	
THERMPAR	2	7850.0	450.0	54.0	0.8	20	
THERMPAR	1000	7850.0	450.0	54.0	0.8	20	
	id	rho (kg/m3)	c (J/kgK)	k (W/mK)	dummy	dummy	
INSIDPAR	1	1.273	0.7	0.0	0	0	
	id	type	K	Emiss			
INSULPAR	10	1	5.0	0.8			

EXPOSURE AND PROTECTION							

	Insulation_ID						
ELMININSUL	10	1121	1222	1323	1424	1525	
		1626	1727	1828	1929	2030	

Figure 2.12-3: FAHTS Control file, *fahts.fem*.

```

HEAD          9 Bay x 3 Storey frame
              U S F O S progressive collapse analysis
              NUS/SINTEF Structural Engineering
              1
TMPLOCON
BANANA
STEELTDEP    ! Temperature dependent material properties
DETEROFF     ! Determinant check switched off
CMAXSTEP     2000
'
'            rest.  xfos  out-file
CSAVE         0      -25    -50
'-----
'            LOAD HISTORY AND DEFORMATION CONTROL
'-----
'            nloads  npostp  mxpstp  mxpdis
CUSFOS        10       100     1.00    0.05
'            lcomb  lfact   mxld    nstep   minstp
'            1      0.1     3.0     50      0.01 ! Dead weight
'            4      0.04    1.0     100     0.01 ! Fire load
'            5      0.04    1.0     100     0.01 ! Fire load
'            6      0.04    1.0     100     0.01 ! Fire load
'            7      0.04    1.0     100     0.01 ! Fire load
'            8      0.04    1.0     100     0.01 ! Fire load
'            9      0.04    1.0     100     0.01 ! Fire load
'            10     0.04    1.0     100     0.01 ! Fire load
'            11     0.04    1.0     100     0.01 ! Fire load
'            12     0.04    1.0     100     0.01 ! Fire load
'            13     0.04    1.0     100     0.01 ! Fire load
'            14     0.04    1.0     100     0.01 ! Fire load
'            15     0.04    1.0     100     0.01 ! Fire load
'            16     0.04    1.0     100     0.01 ! Fire load
'            17     0.04    1.0     100     0.01 ! Fire load
'            18     0.04    1.0     100     0.01 ! Fire load
'            19     0.04    1.0     100     0.01 ! Fire load
'            20     0.04    1.0     100     0.01 ! Fire load
'            21     0.04    1.0     100     0.01 ! Fire load
'            22     0.04    1.0     100     0.01 ! Fire load
'            23     0.04    1.0     100     0.01 ! Fire load
'            ncnods
CNODES        1
'            nodex  idof    dfact
'            35     2       1.
'            35     3       1.
'            36     2       1.
'            36     3       1.
'-----
'            INELASTIC MATERIAL PROPERTIES
'-----
'            matno  E-mod   poiss   yield   density   therm. exp.
MISOIEP       1    205000.0E6  0.3    275.0E6   7850      1.4E-05
MISOIEP       2    205000.0E6  0.3    275.0E6   7850      1.4E-05
'
'            X      Y      Z      Mx      My      Mz
MREF          1000  99     99     99     1002  99   ! Non linear Flexible Connection

```

Figure 2.12-4: USFOS Control file, *usfos.fem*.

```

*****
***** F A H T S *****
*****
FAHTS
Step   Time   Accum.   Max heat   Max   Min
no.    (s)     energy   input      temp. temp.
              (J)      (W/m2)    (degr C) (degr C)
1       36.00  1.761E+05  8.11E+02   20.4   20.0
2       72.00  8.611E+05  1.64E+03   21.6   19.8

199     7164.00  2.301E+09  1.36E+04   799.8  -13.9
200     7200.00  2.292E+09  1.34E+04   795.8  -12.4
===== F A H T S A N A L Y S I S C O M P L E T E D =====

```

Figure 2.12-5: Extract of FAHTS out file, *resf.out*.

2.13 Arched framework exposed to radiation and heat increment.

Key words:

Ex. 13: Steel I-prof Partly exp. Unprotected Radiation Temp. hist.

Material: Steel.

Structure: Arched framework, exposed to both radiation and environmental temperature increment, not protected.

This example shows the use of a combination of user defined temperature histories and radiation.

The structure is modelled as shown in Figure 2.13-1.

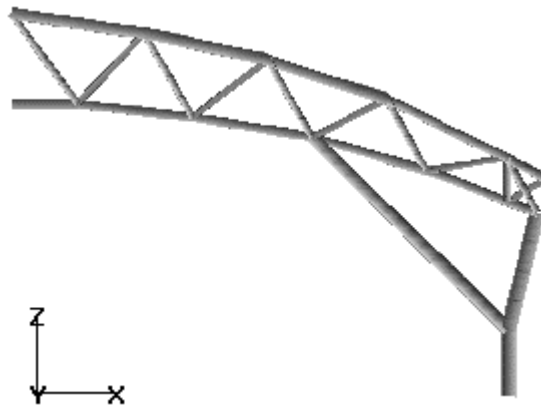


Figure 2.13-1: Structural model.

Yield stress : $\sigma = 275 \text{ MPa}$

Youngs modulus : $E = 205000 \text{ MPa}$

Heat source : User defined radiation and temperature history

Problem:

Structure with both increasing environmental temperature, and radiation.
Several sources.

Solution:

Define all radiation sources like in 2.8, and all temperature sources like in 2.12.
In this example, also the radiation sources are time dependent. This is defined in the same way as temperature fields, by giving a factor for the variation of the intensity over time. The **ref_intensity** factor from USERFIRE is multiplied with the **factor** from TIMEHIST, and results in the total intensity at the given time.

```

HEAD          F A H T S  temperature simulation
              Tuas Checkpoint, SINGAPORE
              Car Fire Simulations

MOVIEPRI
'             N_Lengt  nside  ntop  nbot
MESHIPRO      4         1      2      2
'-----
'             HEAT SOURCE
'-----
'             initemp
INITEMP       30.0
'
'             hist_id   x       y       z       ref_intensity  ref_radius
USERFIRE      1000      11      -1      1         0.2           0.4
USERFIRE      1005      16      -1      1         0.2           0.4
USERFIRE      1010      11       1      1         0.2           0.4
USERFIRE      1010      16       1      1         0.2           0.4
'
'             hist_id field_temp Gas_Abs Lim_Type x1  y1  z1  x2  y2  z2
USERTEMP      100       100       1.0         1      4   -5   0   22   5   20
USERTEMP      100       300.0     1.0         1      8   -5   0   20   5   7
USERTEMP      100       600.0     1.0         1      9   -5   7   18   5   20
'
'             hist_id  Type    time    factor
TIMEHIST      100       1       0.0      0.0
              300.0      0.8
              720.0      1.0
              1980.0     0.7
              2100.0     0.3
              2700.0     0.1
              3600.0     0.0
              7200.0     0.0
TIMEHIST      1000      1       0.0      800.0E3
              300.0      1700.0E3
              720.0      1250.0E3
              1020.0     1400.0E3
              1200.0     1300.0E3
              1500.0     1750.0E3
              1980.0     1000.0E3
              2100.0     500.0E3
              2700.0     250.0E3
              3600.0     150.0E3
              7200.0     100.0E3
TIMEHIST      1005      1       0.0      800.0E3
              300.0      1700.0E3
              720.0      1250.0E3
              1020.0     1400.0E3
              1200.0     1300.0E3
              1500.0     1750.0E3
              1980.0     1000.0E3
              2100.0     500.0E3
              2700.0     250.0E3
              3600.0     150.0E3
              7200.0     100.0E3
TIMEHIST      1010      1       0.0      800.0E3
              300.0      1700.0E3
              720.0      1250.0E3
              1020.0     1400.0E3
              1200.0     1300.0E3
              1500.0     1750.0E3
              1980.0     1000.0E3
              2100.0     500.0E3
              2700.0     250.0E3
              3600.0     150.0E3
              7200.0     100.0E3
'-----
'             ANALYSIS TIME, TEMPERATURE STEPS AND THERMAL PARAMETERS
'-----
'             end-time (s)  nstep    resinc (s)
TEMPSIM       1800.0       50        360.0
'
'             mat_id rho (kg/m3)  c (J/kgK)  k (W/mK)  emiss  convection
THERMPAR      1         7850.0    450.0    54.0     0.8    20
'
'             mat_id rho (kg/m3)  c (J/kgK)  k (W/mK)  dummy  dummy
INSIDPAR      1         1.273    0.7      0.0     0.0    0.0

```

Figure 2.13-2: FAHTS Control file, *fahts.fem*.

```

*****
***** F A H T S *****
*****

```

FAHTS Step no.	Time (s)	Accum. energy (J)	Max heat input (W/m2)	Max temp. (degr C)	Min temp. (degr C)
1	36.00	-1.144E+06	3.25E+03	32.3	28.8
2	72.00	3.323E+06	6.69E+03	38.1	28.0
3	108.00	1.387E+07	1.05E+04	47.6	27.6
4	144.00	3.097E+07	1.49E+04	61.3	27.5
5	180.00	5.523E+07	2.00E+04	79.5	27.7
6	216.00	8.746E+07	2.60E+04	103.1	28.2
7	252.00	1.286E+08	3.31E+04	132.9	29.0
8	288.00	1.799E+08	4.15E+04	169.7	30.1
9	324.00	2.383E+08	4.45E+04	211.5	31.2
10	360.00	2.978E+08	4.40E+04	253.7	32.5
-					
41	1476.00	8.384E+08	1.99E+03	583.5	70.1
42	1512.00	8.321E+08	1.88E+03	579.4	71.1
43	1548.00	8.256E+08	1.62E+03	575.0	71.9
44	1584.00	8.188E+08	1.39E+03	570.4	72.7
45	1620.00	8.118E+08	1.20E+03	565.6	73.4
46	1656.00	8.046E+08	1.05E+03	560.6	74.0
47	1692.00	7.972E+08	9.14E+02	555.5	74.6
48	1728.00	7.897E+08	7.95E+02	550.3	75.0
49	1764.00	7.821E+08	6.85E+02	545.1	75.4
50	1800.00	7.743E+08	5.80E+02	539.8	75.7

```

===== F A H T S   A N A L Y S I S   C O M P L E T E D =====

```

Figure 2.13-3: Extract of result file from FAHTS analysis, *resf.fem*.

2.14 Single frame exposed to pool fire placed between columns.

Key words:

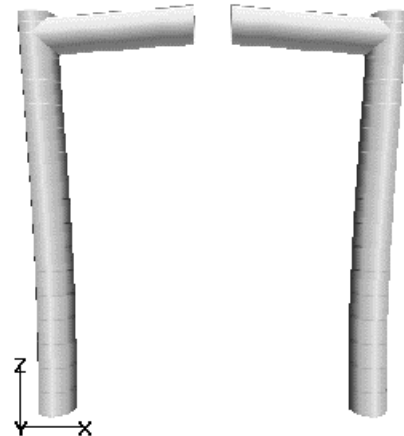
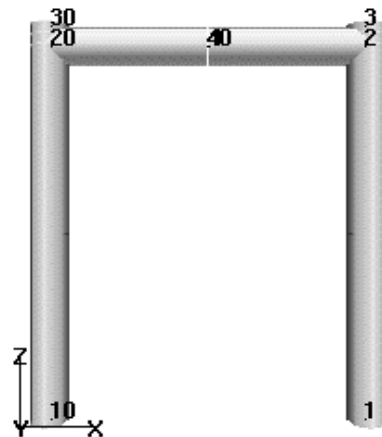
Ex. 14: Steel Pipe Fully exp. Unprotected Kameleon Static ana.

Material: Steel.

Structure: Single frame, node load in node 40, full exposure, unprotected.
The two horizontal elements are not connected at the middle.

This example shows structures with thermal expansion.

The frame is modelled with nodes as shown in Figure 2.14-1.



Analysis model; Time=180

Figure 2.14-1: Structural model with node and element numbers.

Figure 2.14-2: Deformed model.

Column length	:	9.990 m
Diameter (outer)	:	1.000 m
Thickness	:	0.020 m
Beam length	:	7.960 m
Diameter (outer)	:	1.000 m
Thickness	:	0.020 m
Yield stress	:	$\sigma = 248 \text{ MPa}$
Youngs modulus	:	$E = 210000 \text{ MPa}$
Fire load	:	From FIREINT database

Problem:

Beam not connected at the middle.

Solution:

Four nodes; 20, 40, 4 and 2 defines the beam. A beam not connected at the middle will have to be composed of the two elements 20-40 and 4-2. There is no connection between the nodes 40 and 4, and the beam is therefore not connected at the middle. The frame will be free to expand because of the increasing temperature, and will not obtain any inner forces or stress.

HEAD

SESAM

NODAL DATA

	node-id	x	y	z	boun.cond.					
					ix	iy	iz	irx	iry	irz
NODE	10	-3.99	0.0	0.01	1	1	1	1	1	1
NODE	20	-3.99	0.0	9.45						
NODE	30	-3.99	0.0	9.99						
NODE	40	-0.01	0.0	9.45						
NODE	1	3.99	0.0	0.01	1	1	1	1	1	1
NODE	2	3.99	0.0	9.45						
NODE	3	3.99	0.0	9.99						
NODE	4	0.01	0.0	9.45						
	load-case	node-no	Fx	Fy	Fz	Mx	My	Mz		
NODELOAD	1	40	0.0	0.0	-1000.0					

ELEMENT DATA

	elem-ID	np1	np2	mater	geom	lcoor
BEAM	1020	10	20	10000	100	1
BEAM	2030	20	30	10000	100	1
BEAM	2040	20	40	10000	100	1
BEAM	12	1	2	10000	100	1
BEAM	23	2	3	10000	100	1
BEAM	24	2	4	10000	100	1
	lco-no	dx	dy	dz		
UNITVEC	1	-1000.0	0.0	100.0		
REFINE	8	1020	12			
REFINE	5	2040	24			

CROSS-SECTION DATA

	geo_no	Do	Thick
PIPE	100	1.000	0.020

MATERIAL PROPERTIES

	mat_no	E-mod	Poiss
MISOSEL	10000	210000.0E6	0.3

Figure 2.14-3: Structure file, *stru.fem*.

```

HEAD          F A H T S  temperature simulation
              Beam + Column
              TUBE - profile
SHAPFACT      ! Shape factor calc. is switched on.
,
MOVIEPRI      ! Override default mesh refinement.
-----
              TYPE OF FIRE AND EXPOSURE
-----
FIREINT       ! Fire scenario from FIREINT database.
,
              initemp
INITEMP       10.0
-----
              ANALYSIS TIME AND TEMPERATURE STEPS
-----
              end-time (s)  nstep      resinc (s)
TEMPSIM       3600.0      100        360.0
-----
              THERMAL PARAMETERS OF MATERIAL AND INSULATION
-----
              id    rho(kg/m3)  c (J/kgK)  k (W/mK)  emiss
THERMPAR      10000    7850.0    500.0    50.0    0.8
,
              id    rho(kg/m3)  c (J/kgK)
INSIDPAR       1       1.273    0.7

```

Figure 2.14-4: FAHTS Control file, *fahts.fem*.

```

HEAD          FAHTS example : Simple tubular frame, no connection mid span
              U S F O S
              SINTEF div of Structural Engineering
,
-----
              AMOUNT OF INPUT AND OUTPUT PRINTED TO FILE
-----
              inpri    outpri    termpri
CPRINT        1        1        1
-----
              LOAD HISTORY AND DEFORMATION CONTROL
-----
              nloads    npostp    mxpstp    mxpdis
CUSFOS        30        15        1.00     0.05
,
              lcomb     lfact     mxld      nstep     minstp
              4         0.5      1.0       15        0.05
              5         0.5      1.0       15        0.05
              6         0.5      1.0       15        0.05
              7         0.5      1.0       15        0.05
              8         0.5      1.0       15        0.05
              9         0.5      1.0       15        0.05
              10        0.5      1.0       15        0.05
              11        0.5      1.0       15        0.05
              12        0.5      1.0       15        0.05
              13        0.5      1.0       15        0.05
,
              ncnodes
CNODES        1
,
              nodex     idof      dfact
              40        3        1.
,
-----
              MATERIAL PROPERTIES
-----
              matno     E-mod     poiss     yield     density    therm.exp.
MISOIEP       10000    210000.0E6  0.3      248.E6     7850.0     1.4E-05

```

Figure 2.14-5: USFOS Control file, *usfos.fem*.

***** F A H T S *****					
FAHTS Step no.	Time (s)	Accum. energy (J)	Max heat input (W/m2)	Max temp. (degr C)	Min temp. (degr C)
1	36.00	1.194E+05	2.00E+03	10.9	9.7
2	72.00	2.384E+05	2.00E+03	11.8	9.5
3	108.00	3.567E+05	1.99E+03	12.6	9.3
-					
98	3528.00	1.527E+08	2.94E+03	186.9	8.7
99	3564.00	1.536E+08	2.93E+03	187.4	8.8
100	3600.00	1.546E+08	2.91E+03	187.9	8.9

Figure 2.14-6: Extract of FAHTS out file, *resf.out*.



Figure 2.14-7: The global displacement's development over time.

2.15 Single frame exposed to pool fire placed between columns.

Key words:

Ex. 15: Steel Pipe Fully exp. Unprotected Kameleon Temp. sim.

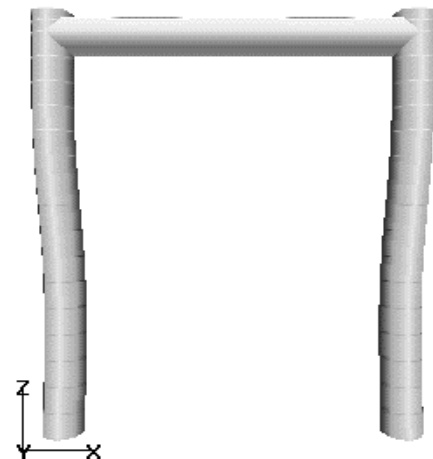
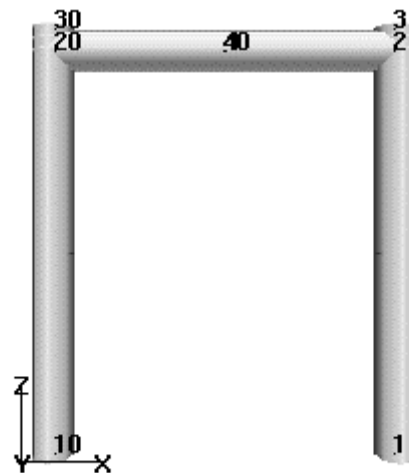
Material: Steel.

Structure: Single frame, full exposure, unprotected.

Similar structure as in the previous example, except from the two horizontal elements, which are connected at the middle.

This example shows structures with thermal expansion.

The frame is modelled with nodes as shown in Figure 2.15-1.



Analysis model; Time=180

Figure 2.15-1: Structural model with node and element numbers.

Figure 2.15-2: Deformed model.

Problem:

Simple frame, beam connected at the middle.

Solution:

Define elements between the nodes 2 and 40 instead of 2 and 4. This gives a continuous beam.

When the beam is connected at the middle, the frame will not be able to expand freely. This will lead to great inner forces and stress in the frame.

ELEMENT DATA								
	elem-ID	np1	np2	mater	geom	lcoor	ecc1	ecc2
BEAM	1020	10	20	10000	100	1		
BEAM	2030	20	30	10000	100	1		
BEAM	2040	20	40	10000	100	1		
BEAM	12	1	2	10000	100	1		
BEAM	23	2	3	10000	100	1		
BEAM	24	2	40	10000	100	1		

Figure 2.15-3: Structure file for USFOS and FAHTS, *stru.fem*.

***** F A H T S *****						
FAHTS Step no.	Time (s)	Accum. energy (J)	Max heat input (W/m2)	Max temp. (degr C)	Min temp. (degr C)	
1	36.00	1.861E+06	1.80E+04	14.8	9.8	
2	72.00	5.390E+06	1.78E+04	24.4	9.6	
3	108.00	8.876E+06	1.74E+04	33.5	9.4	
-						
98	3528.00	1.794E+08	3.19E+03	195.4	9.9	
99	3564.00	1.804E+08	3.17E+03	195.8	9.9	
100	3600.00	1.814E+08	3.16E+03	196.2	9.9	

Figure 2.15-4: Extract of FAHTS out file, *resf.out*.

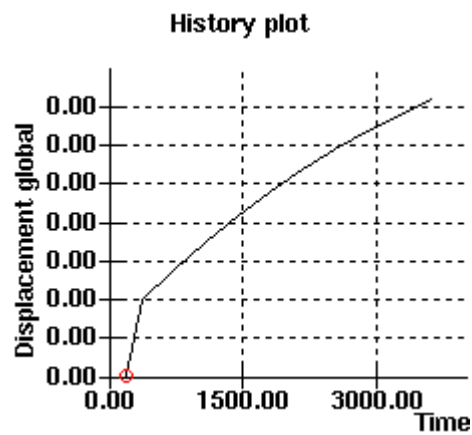


Figure 2.15-5: The global displacement's development over time.