LAB 1. Finite element analysis of heat transfer and thermal stress in a 3D pipe model.

CASE 1.

<u>Thermal analysis in a steady state. Applied temperatures on the inner and outer surfaces.</u>

1. Geometry of a quarter of the thick walled pipe model (Fig. 1).
Create a cylinder sector: the outer radius b = 40 mm, the inner radius a = 30 mm, the height c = 50 mm, the sector ending angle 90° (*Preprocessor>Modeling>Create>Volumes>Cylinder>By Dimensions)*

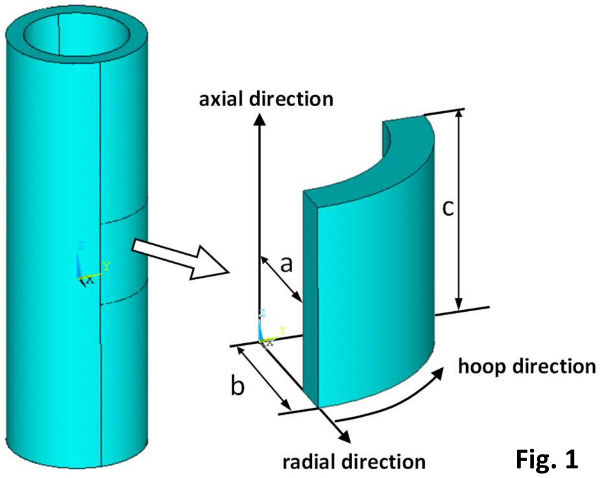


Fig. 1

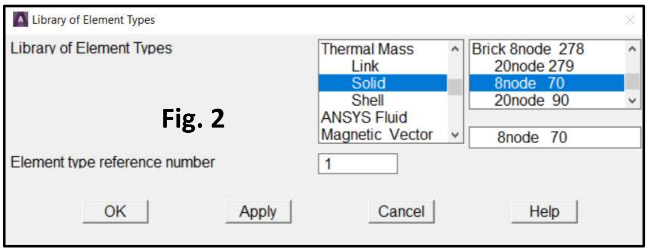


Fig. 2

2. Choose SOLID70 finite element: (brick 8-node) (*Preprocessor>Element Type>Add/Edit/Delete*) (Fig.2)
3. Define thermal and structural material properties of a steel (linear elastic and isotropic): (*Preprocessor>Material Props>Material Models)* (Fig. 3)

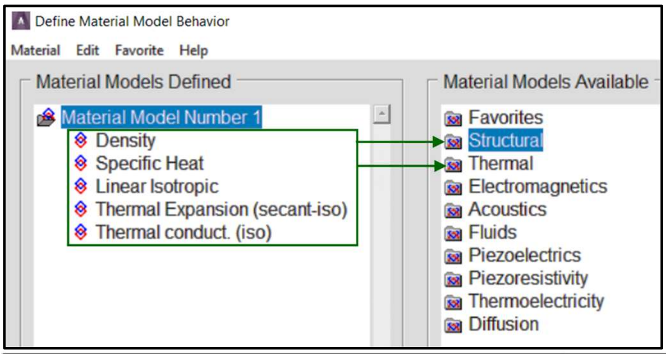


Fig. 3

Thermo-mechanical properties	**a, b, c (m) SI system of units**	**a, b, c (mm)**
Density[1]	**7800 (kg/m^3)**	**7.8E-9 (Ns^2/mm^4)**
Specific Heat[1]	**452 (J/kgK)**	**4.52E8 (mm^2/s^{2}K)**
Linear Isotropic: Young's modulus, Poisson's ratio	**2E11 (Pa) 0.3**	**2E5 (MPa) 0.3**
Thermal expansion (secant-iso): secant coefficient of thermal expansion reference temperature (zero thermal strain)	**1.2E-5 (1/K) 0 (°C)**	**1.2E-5 (1/K) 0 (°C)**
Thermal conductivity	**50 (W/mK)**	**50 (mW/mmK)**
[1] must be defined for transient heat flow		

4. Specify density of discretization on lines and create a mapped hexahedral mesh (Fig. 4)
5. Save the mesh as an image (*Plot> Elements, PlotCtrls> Redirect Plots-> To JPEG File …*)

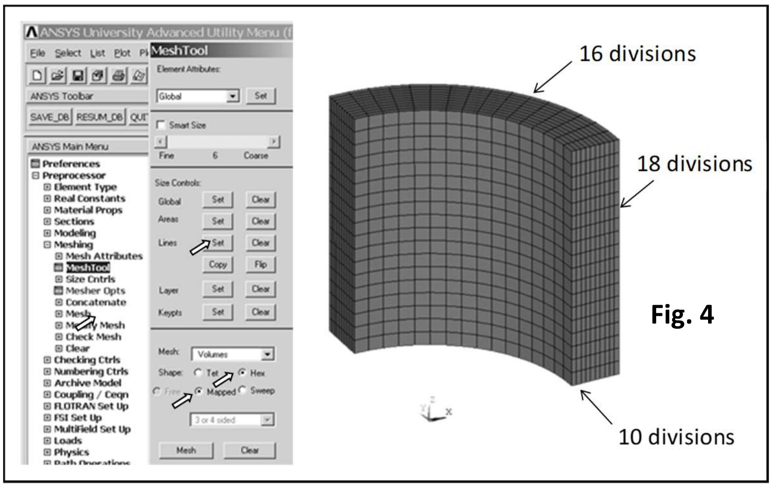


Fig. 4

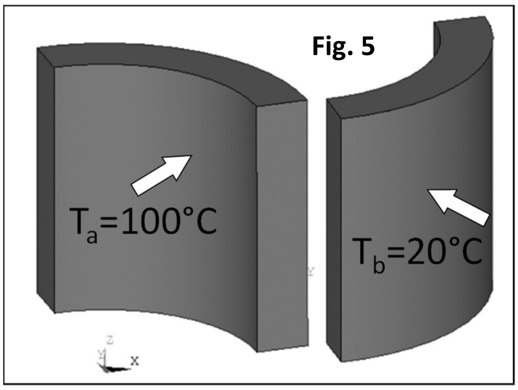


Fig. 5

6. Save the database (*Utility Menu>File>Save As…*, for example: pipe_thermal.db)
7. Apply boundary conditions: temperatures on the inner and outer surfaces (Fig. 5), (*Solution> Apply> Thermal> Temperature> On Areas*)
8. Solve (*Solution-> Solve-> Current LS*)
9. Read thermal analysis results (*Main Menu> General Postproc> Read Results> First Step*)
10. Choose the Global Cylindrical system for the results presentation (Main Menu> General Postproc> Options for Output> [RSYS]-> Global Cylindric)
11. Plot a contour map of the temperature (*Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu>DOF Solution>Nodal temperature*). Save it as an image (Fig. 6).
12. Plot a contour map of the thermal gradient in the radial direction (*Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu> Thermal Gradient>X-Component of thermal gradient*). Save it as an image (Fig. 7).
13. Plot a contour map of the thermal flux in the radial direction (*Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu >Thermal Gradient>X-Component of thermal flux*). Save it as an image (Fig. 8).
14. Plot a graph of the temperature as a function of the distance measured along the radial coordinate:
· select 2 nodes in locations 'MX' and 'MN' (shown in Fig. 6) (*Main Menu>General Postproc>Path Operations>Define Path>By Nodes* (ok), Name= path1, nSets =30, nDiv=20)
· select temperature (*Main Menu>General Postproc>Path Operations>Map onto Path* (ok) – the field 'Lab' can be left blank)
· choose the temperature item to plot (*Main Menu>General Postproc>Path Operations>Plot Path Item>On Graph*). Save the graph as an image (Fig. 9).
15. Fill Table 1 (numbers & units).

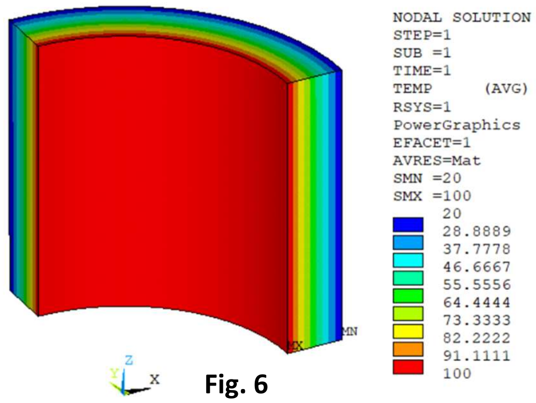


Fig. 6

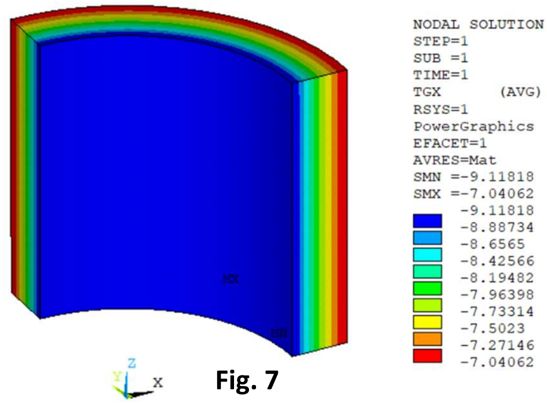


Fig. 7

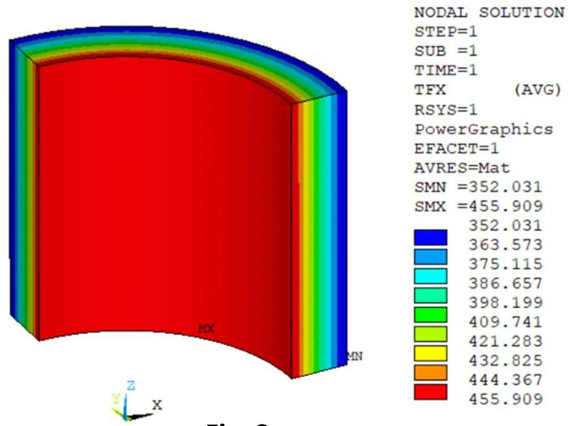


Fig. 8

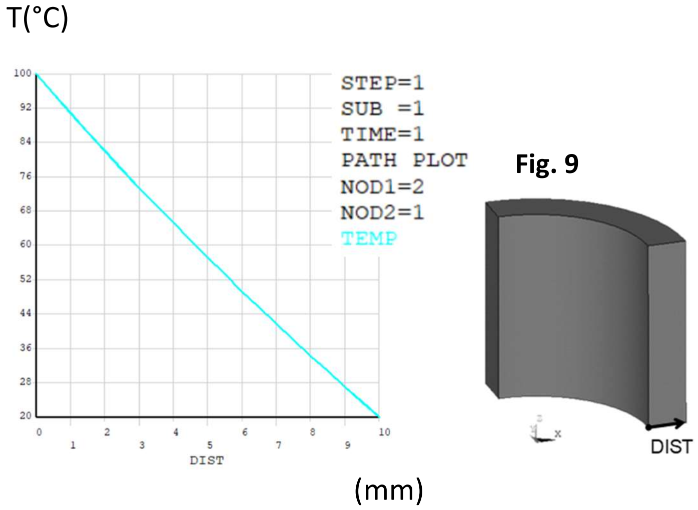


Fig. 9

Analytical solution

$$\text{temperature: } T(r) = T_a + \frac{T_b - T_a}{ln\,\left(\frac{b}{a}\right)}\ ln\left(\frac{r}{a}\right)$$

<u>Structural thermal-stress analysis of the pipe with constrained ends. Temperature distribution obtained for temperatures applied on the inner and outer surfaces.</u>

16. Switch the element type from thermal to structural (*Preprocessor>Element Type>Switch Elem Type*) (Fig.10) and adjust SOLID 185 options to a linear static analysis requirements (*Preprocessor>Element Type>Add/Edit/Delete> Options*) (Fig. 11).

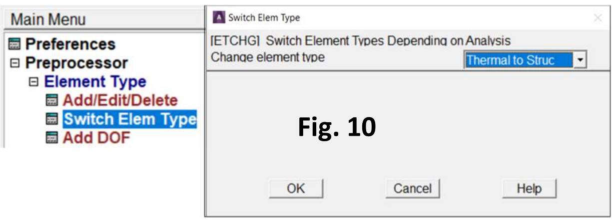


Fig. 10

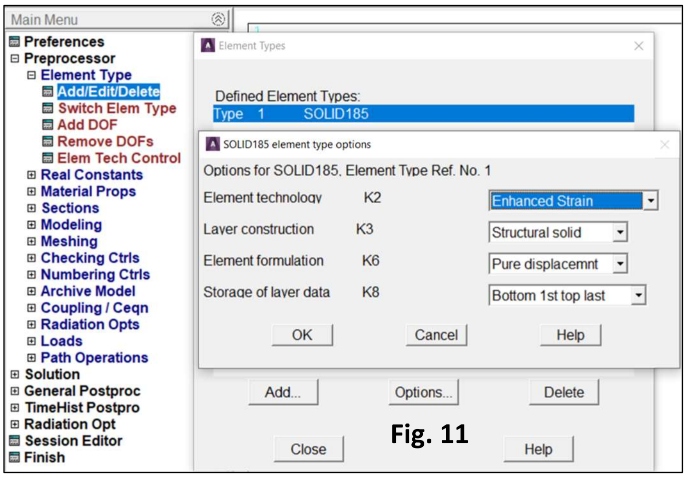


Fig. 11

17. Apply structural symmetry boundary conditions on 4 cross-sectional areas (*Solution> Define Loads>Apply> Structural> Displacement> Symmetry B.C.> On Areas*) (Fig. 12)
18. Apply temperatures from the thermal analysis (*Solution> Define Loads>Apply> Structural> Temperature> From Therm An*) (Fig. 13). In this example, the jobname is ‘file’. Extension ‘rth’ is related to the results stored in the Working Directory as nodal temperatures.
19. Plot elements and check the temperatures (*Utility Menu> Plot> Elements and Utility Menu>PlotCtrls> Symbols>Body Load Symbols> Structural Temps*)

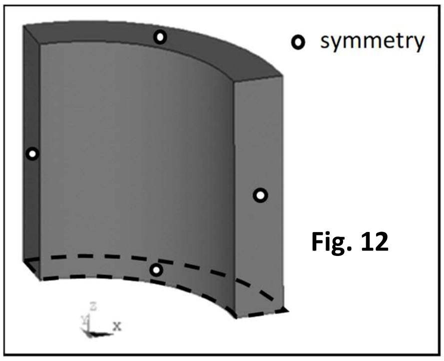


Fig. 12

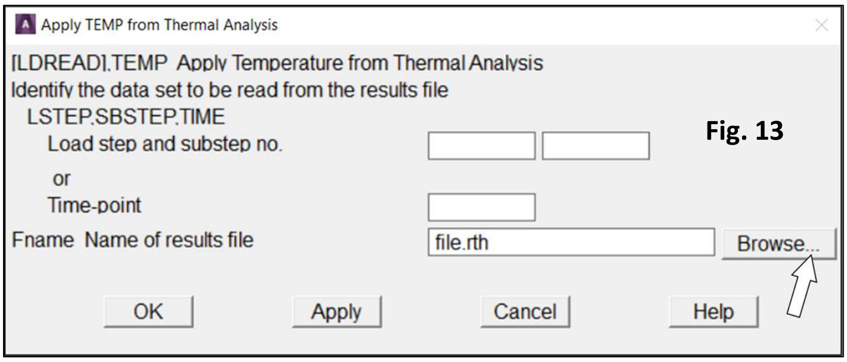


Fig. 13

20. Solve (*Solution-> Solve-> Current LS*)
21. Read structural analysis results (*Main Menu> General Postproc> Read Results> First Step*)
22. Choose the Global Cylindrical system for the results presentation (*Main Menu> General Postproc> Options for Output> [RSYS]-> Global Cylindric*)
23. Plot and save as images the contour maps of stress components: SX (radial), SY (hoop), SZ (axial), and equivalent Von Mises stress (*Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu>Stress…*) (Fig. 14-17)

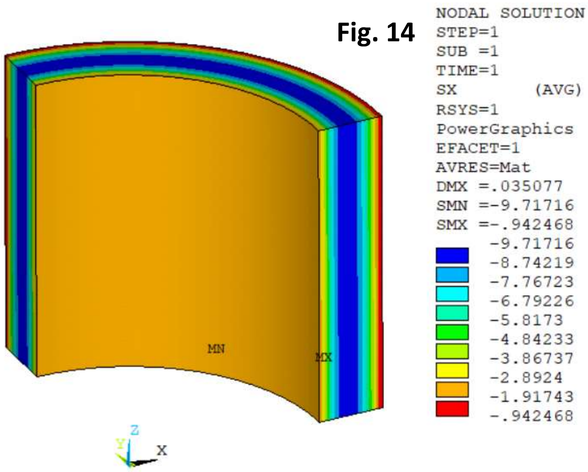


Fig. 14

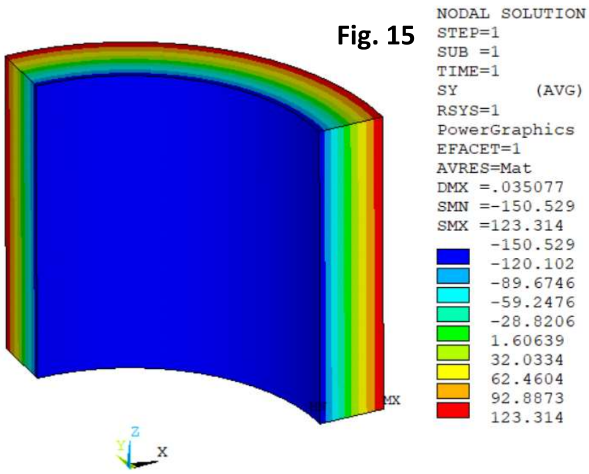


Fig. 15

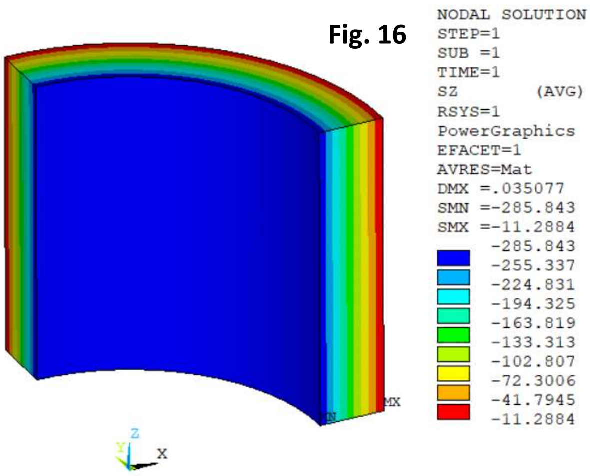


Fig. 16

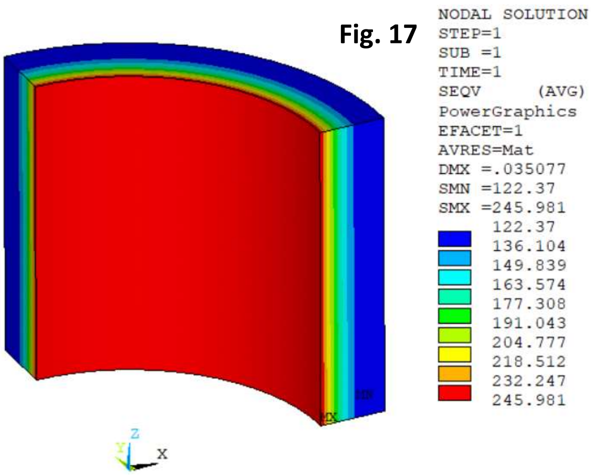


Fig. 17

24. Plot a graph of stress components as a function of the distance measured along the radial coordinate and save it as an image (Fig. 18).

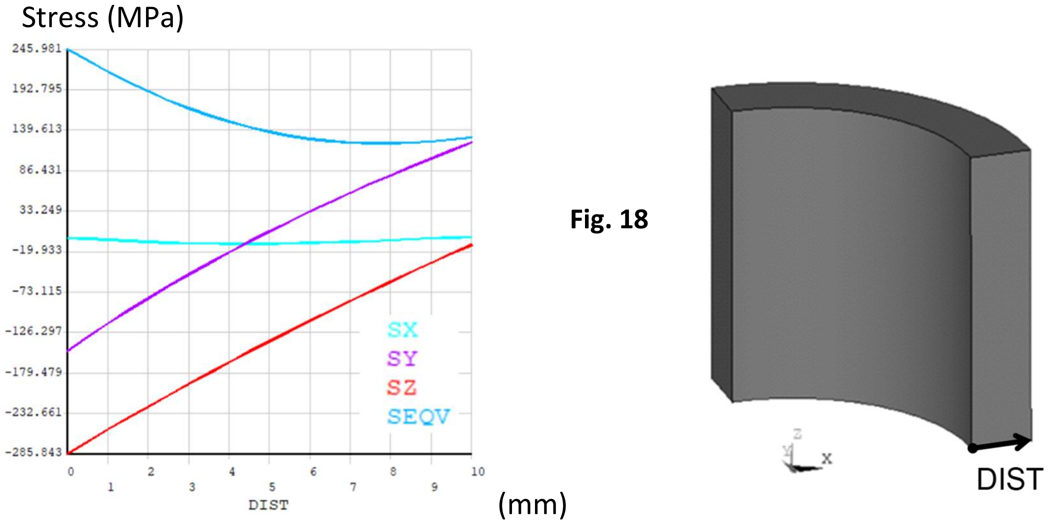


Fig. 18

25. Fill Table 2 (numbers & units).

Analytical solution

radial stress: $\sigma_r(r) = C\left(\frac{\ln(b/r)}{\ln(b/a)} - \frac{b^2/r^2-1}{b^2/a^2-1}\right)$; $\quad C = \frac{-E\alpha(T_a-T_b)}{2(1-\nu)}$

hoop stress: $\sigma_h(r) = C\left(\frac{\ln(b/r)-1}{\ln(b/a)} + \frac{b^2/r^2+1}{b^2/a^2-1}\right)$

axial stress: $\sigma_z(r) = \nu\big(\sigma_r(r) + \sigma_h(r)\big) - \alpha E T(r)$

equivalent Von Mises stress: $\sigma_{eqv}(r) = \sqrt{\frac{1}{2}\left[\big(\sigma_r(r) - \sigma_h(r)\big)^2 + \big(\sigma_h(r) - \sigma_z(r)\big)^2 + \big(\sigma_r(r) - \sigma_z(r)\big)^2\right]}$

CASE 2.

The scope is to consider the convection boundary conditions in the thermal analysis. Assumptions:

- thermal analysis of the pipe with the convection boundary condition applied on the inner and outer surfaces (*Solution>Define Loads>Apply>Thermal>Convection> On Areas*) (Fig.19). Delete any applied temperatures before.
- structural thermal-stress analysis of the pipe with constrained ends (top and bottom surfaces).

Save graphs of temperature and stress components along radius as images and fill Tables 3 and 4.

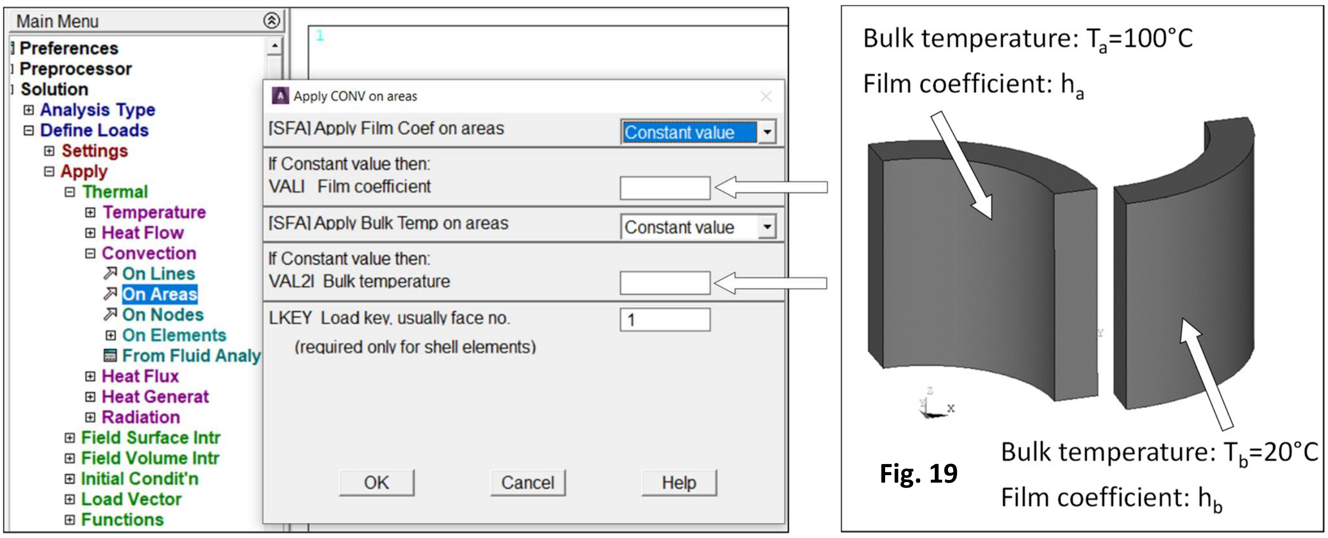


Fig. 19

CASE 3.

The scope is to check the effect of compensation of thermal elongation on a stress in the pipe.
Assumptions:

- temperature distribution from CASE 2 (convection boundary condition),
- structural thermal-stress analysis of the pipe with compensation of thermal elongation. This is done by coupling nodes belonging to the top surfaces instead of applying the symmetry boundary condition (*Main Menu> Preprocessor>Coupling/Ceqn> Couple DOFs*) (Fig. 20).

Save the graph of stress components along radius as image and fill Table 5.

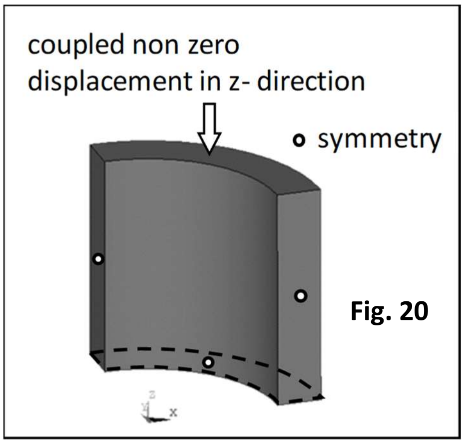
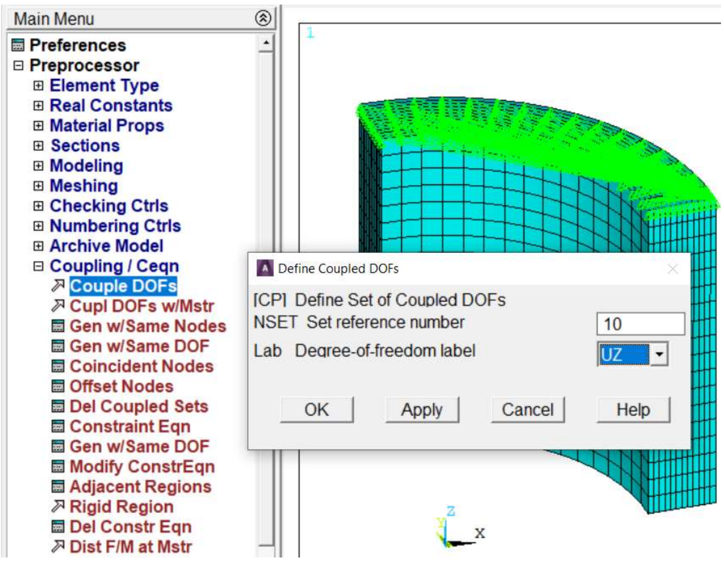


Fig. 20



CASE 4.

The scope is to check the effect of insulation on temperature and stress distributions.

Assumptions:

- thermal analysis of the insulated pipe (Fig. 21) with the convection boundary condition applied on the inner surface of the pipe and the outer surface of the insulation (the same values of *T* and *h* as shown in Fig. 19).
- the two parts are glued (ideal thermal contact).
- structural thermal-stress analysis of the pipe with compensation of thermal elongation.

Save graphs of temperature (in the entire model) and stress components (in the pipe) along radius as images and fill Tables 6 and 7.

Thermo-mechanical properties of insulation	a, b, c, f (m) SI system of units	a, b, c, f (mm)
Linear Isotropic: Young's modulus, Poisson's ratio	1E9 (Pa) 0.35	1000 (MPa) 0.35
Thermal expansion (secant-iso): secant coefficient of thermal expansion reference temperature (zero thermal strain)	1.2E-5 (1/K) 0 (°C)	1.2E-5 (1/K) 0 (°C)
Thermal conductivity	0.1 (W/mK)	0.1 (mW/mmK)

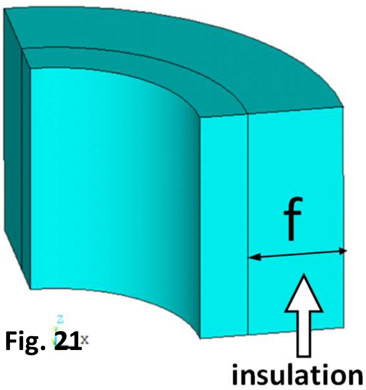


Fig. 21

CASE 5.

The scope is to consider a transient heat flow in the pipe.

Assumptions:

- In the initial state (time t = 0) the temperature in the pipe is uniform (20°C).
- thermal analysis of the pipe with the convection boundary condition applied on the inner and outer surfaces.
- static structural analysis of the pipe based on results of a transient thermal analysis,

1. Prepare the initial step using the one of two possible ways:
a) set a uniform temperature to 20°C (Main Menu>Solution>Define Loads>Apply>Thermal>Temperature> Uniform Temp) (initial state is defined immediately),
b) perform a thermal analysis in the steady state, assuming temperatures T_a = T_b = 20°C (see the first part of CASE1). Replace the name of resulting file '*.rth' with 'initial_temp.rth' (this way is useful if the temperature distribution in the initial step is nonuniform).
2. Choose a transient analysis (Fig.22) and a full solution method.
3. Delete temperatures on areas and apply convection on the inner and outer surfaces (see Fig. 19).

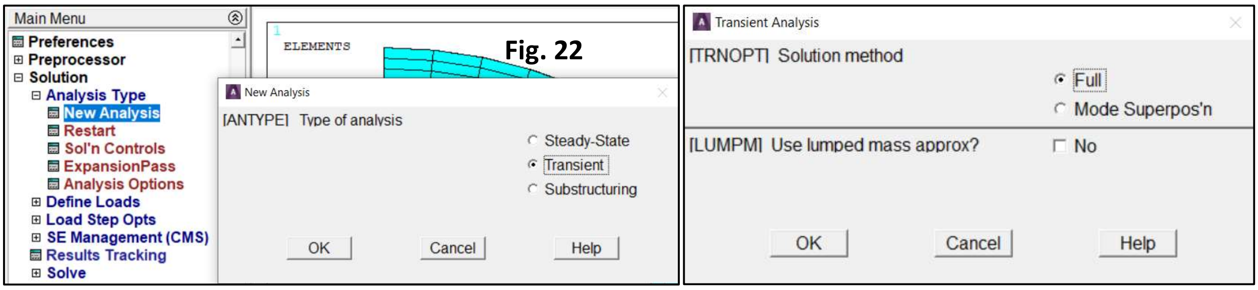


Fig. 22

4. Read the temperature distribution in the initial state (Fig. 23) (skip this point if the uniform temperature was defined)

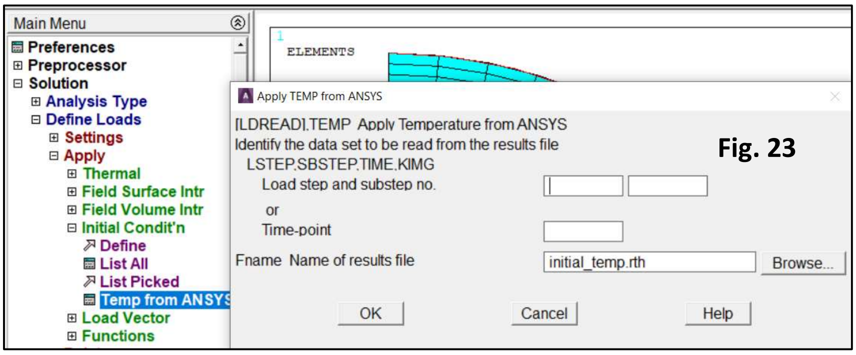


Fig. 23

5. Set the analysis options and create loadsteps.

We assume that a hot medium (T_a = 100°C, h_a) appears in the pipe immediately, so that the convection boundary condition is applied as 'stepped' and remains unchanged during the analysis. An automatic time stepping is usually used if a nonlinear analysis is not converged, so the system changes the time step to improve the convergence. This is not our case, so we turn it off to get the results in chosen time moments set by 'Time and Substeps Options' (Fig. 24 and 25).

The transient thermal analysis is performed in a sequence of two loadsteps (Fig. 24) :

- first, to capture the thermal shock phenomenon (the greatest temperature difference between the inner and outer surfaces) - specify TIME, NSUBST, FREQ and write the loadstep 1 (Fig. 26),
- second, to reach the steady state - specify TIME, NSUBST, FREQ and write the loadstep 2.

6. Solve (*Solution-> Solve->* ***From LS Files****;* LSMIN=1, LSMAX=2, LSINC=1*)*

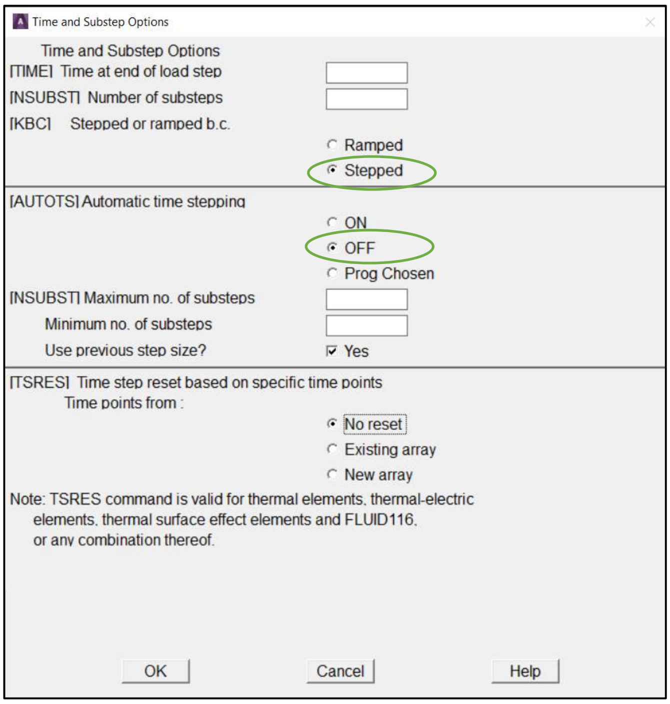


Fig. 24

Load step (LSTEP)	Time at end of loadstep (TIME) (s)	Number of substeps (NSUBST)
1	10	20
2	1800	30

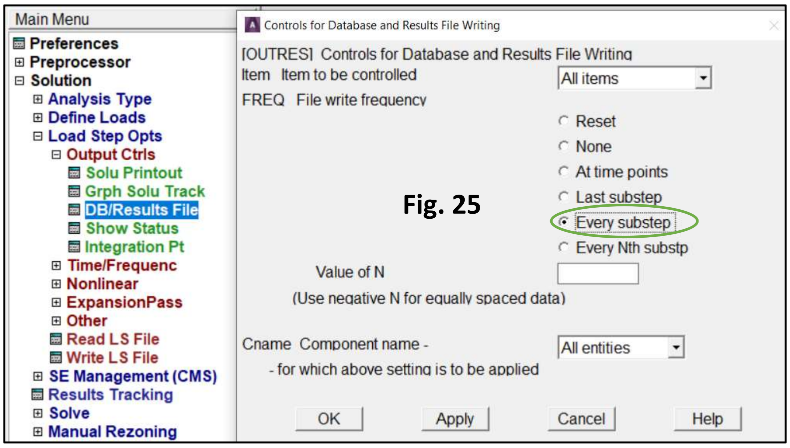


Fig. 25

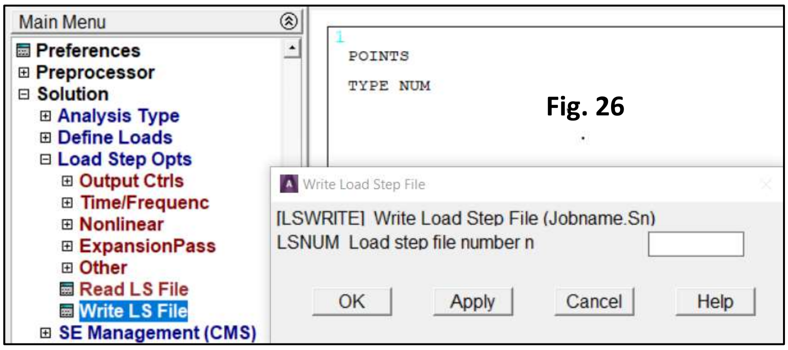


Fig. 26

7. Read temperatures of the inner and outer surfaces as a function of time, first T_a then T_b (*Main Menu>TimeHist Postpro>Variable Viewer*) (Fig. 27).
8. Plot $T_a(t)$ and $T_b(t)$ on a graph (Fig. 28) and save the graph as an image.
9. Define a new variable delta_T(t) = $T_a(t)$ - $T_b(t)$ (Fig. 29)
10. Close and reopen the Variable Viewer.
11. Set a logarithmic scale for the time axis (*Utility Menu>PlotCtrls>Style>Graphs>Modify Axes…> X-Axis Scale->Logarithmic*). Plot and save the graph of delta_T(t) (Fig. 30). Check the time of delta_T max (Fig. 31).

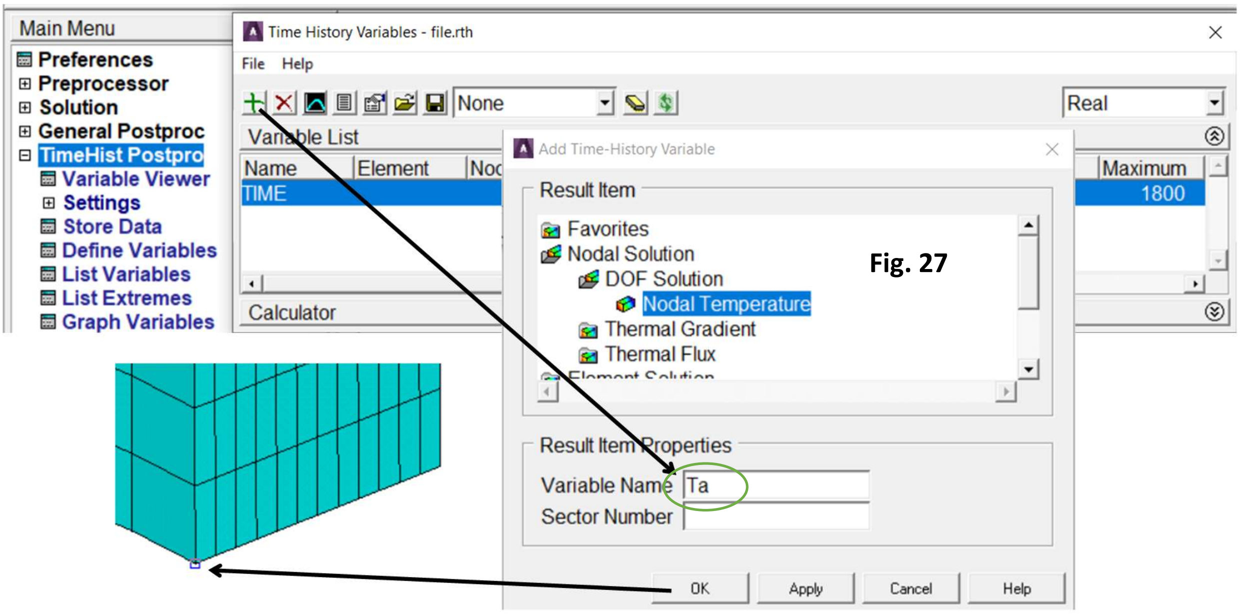


Fig. 27

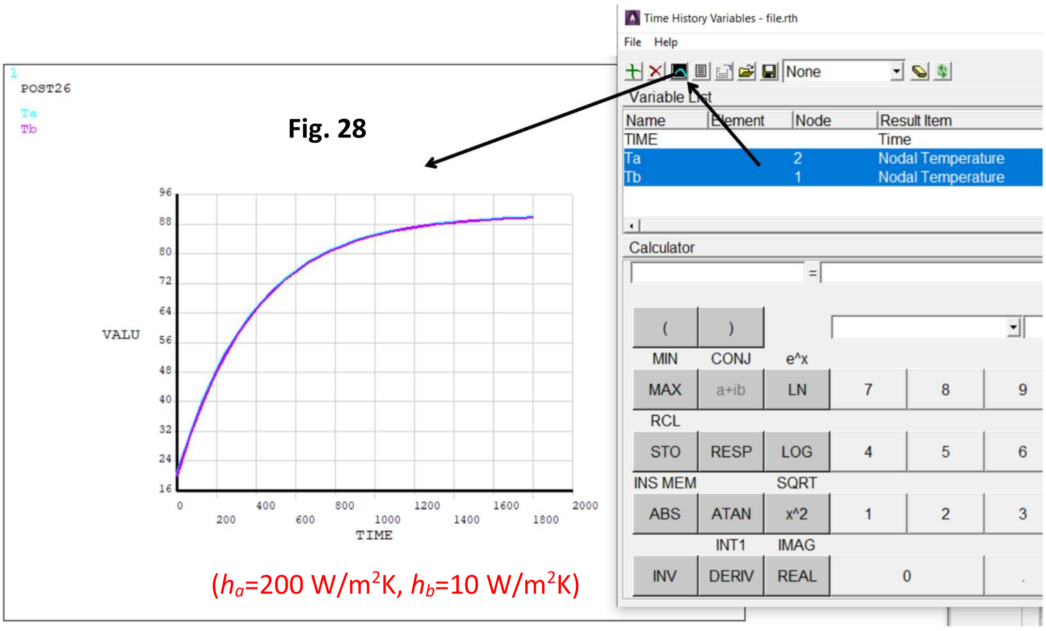


Fig. 28

(h_a=200 W/m^{2}K, h_b=10 W/m^{2}K)

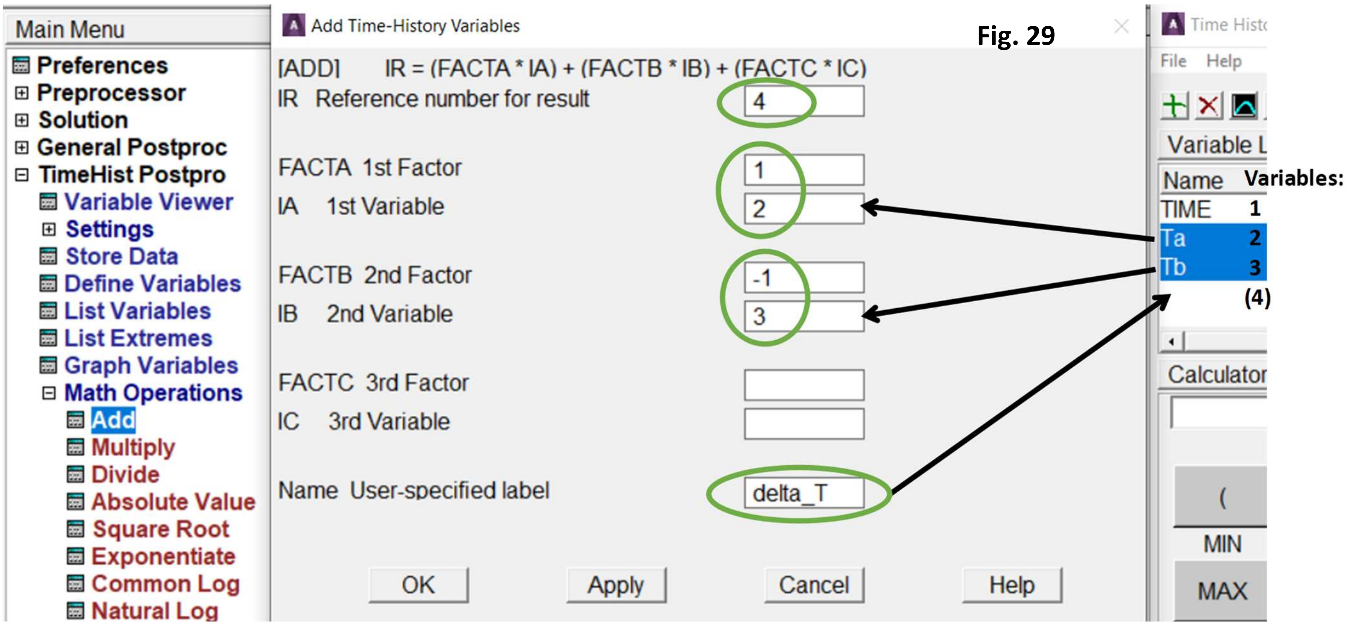


Fig. 29

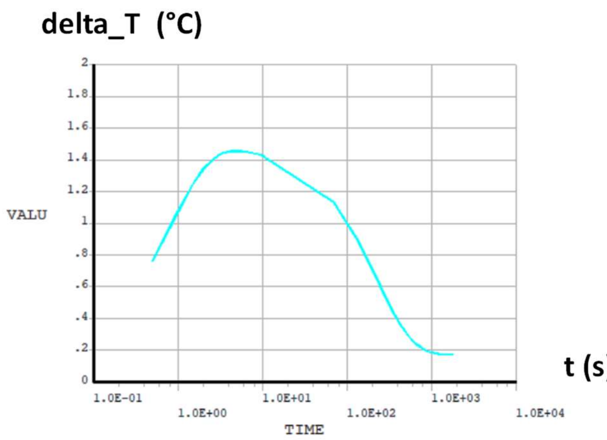


Fig. 30

(h_a=200 W/m^{2}K, h_b=10 W/m^{2}K)

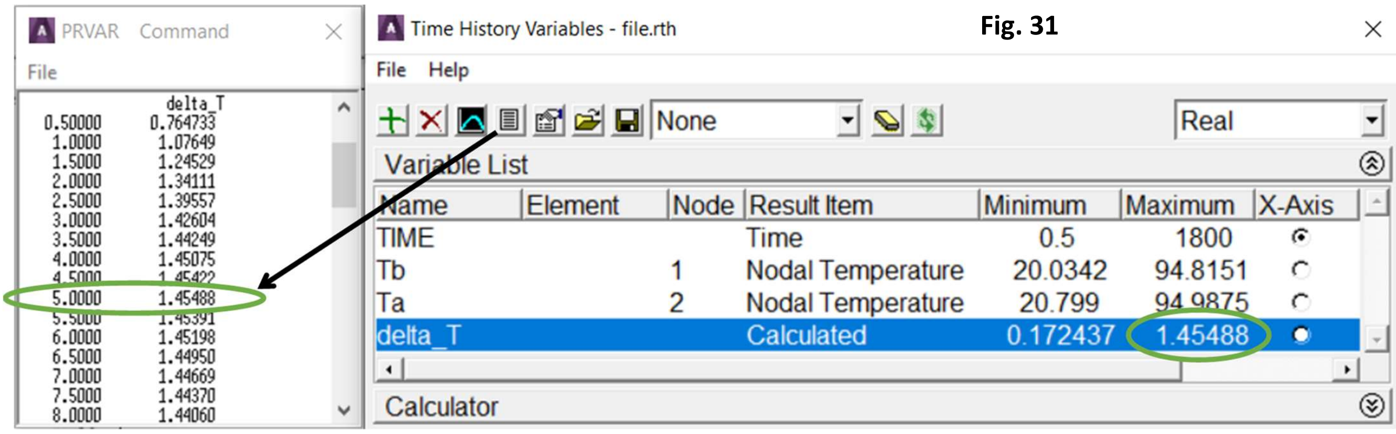


Fig. 31

12. Switch the element type from thermal to structural (*Preprocessor>Element Type>Switch Elem Type*) (Fig.10) and adjust SOLID 185 options to a linear static analysis requirements (*Preprocessor>Element Type>Add/Edit/Delete> Options*) (Fig. 11)
13. Apply structural boundary conditions and coupling of the nodes located on the top surface (Fig. 20)
14. Create a text file and save it as *macro.txt* in the working directory.

```
! _______________MACRO SOLVES STRUCTURAL MODEL OF PIPE _________________
! ________WITH TEMPERATURES FROM TRANSIENT THERMAL ANALYSIS___________
!*
/SOLU                                  !  Main Menu>Solution
ANTYPE,0                               !  a static structural analysis
*DO,index1,1,20,1                      !  20 – number of substeps in loadstep 1
TIME,index1*10/20                      !  10 – time at end of loadstep 1
NSUBST,1,,,                            !  1 – one substep
LDREAD,TEMP,1, index1, , ,'','rth',' ' !  1 – loadstep 1
SOLVE
*ENDDO
*DO,index2,1,30,1                      !  30 – number of substeps in loadstep 2
TIME,index2*(1800-10)/30+10            !  10, 1800 – times at end of loadsteps 1 and 2
NSUBST,1,,,                            !  1 – one substep
LDREAD,TEMP,2, index2, , ,'','rth',' ' !  2 – loadstep 2
SOLVE
*ENDDO
!* ___________________________END OF MACRO _____________________________
```

15. Execute the macro (*Utility Menu>File>Read Input from… -> macro.txt)*
16. Read the values of equivalent von Mises stress in selected nodes (Fig. 32) as a function of time.
17. List stress values (Fig. 33). Check the time at which the Von Mises stress reaches maximum (Fig. 33) and plot and save the graph as an image (logarithmic scale for time) (Fig. 34). Fill the time value in Table 8.

Fig. 32

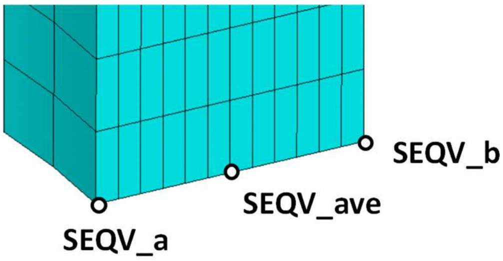
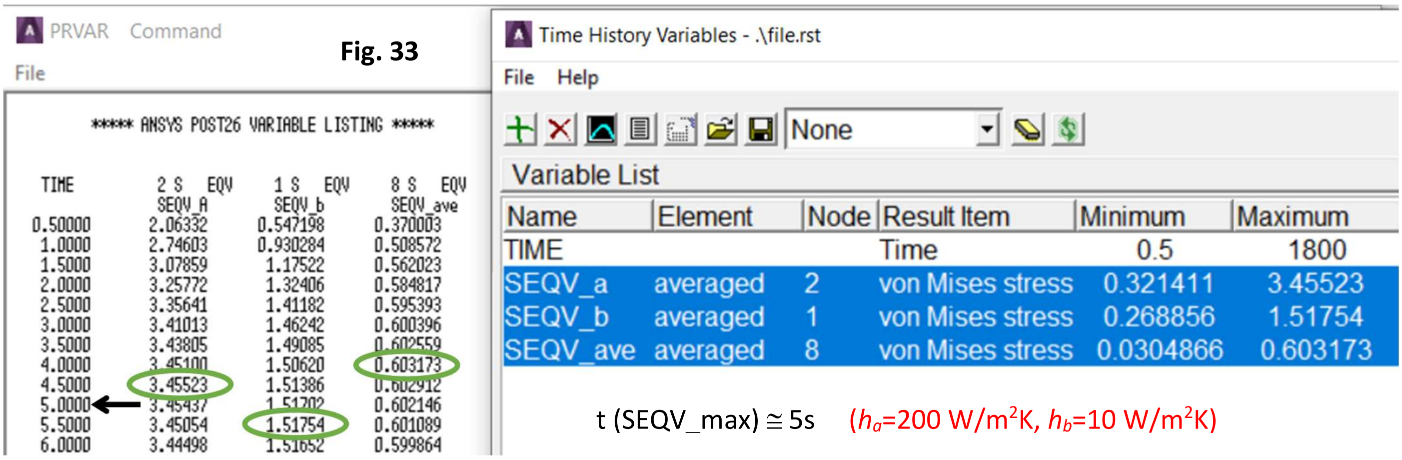


Fig. 33



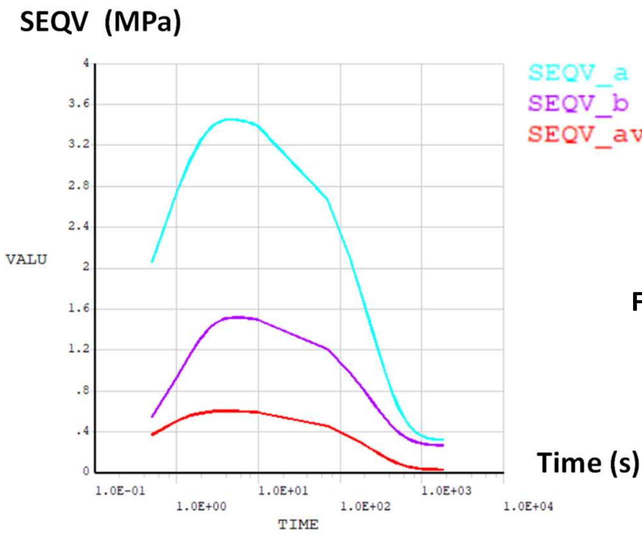


Fig. 34

18. Read results for the time at which stress reaches maximum (*Main Menu>General Postproc>Read Results>By Pick*) (Fig. 35). Plot and save as images the contour maps of:
- stress components: SX (radial), SY (hoop), SZ (axial), and equivalent Von Mises stress (SEQV) (*Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu>Stress…*),
- temperature and the axial displacement (*Main Menu>General Postproc>Plot Results>Contour Plot>DOF Solution>Body temperatures,* and *Z- Component of displacement*).
19. Read results for time = 1800s and repeat results processing from point 17. Fill Table 8.

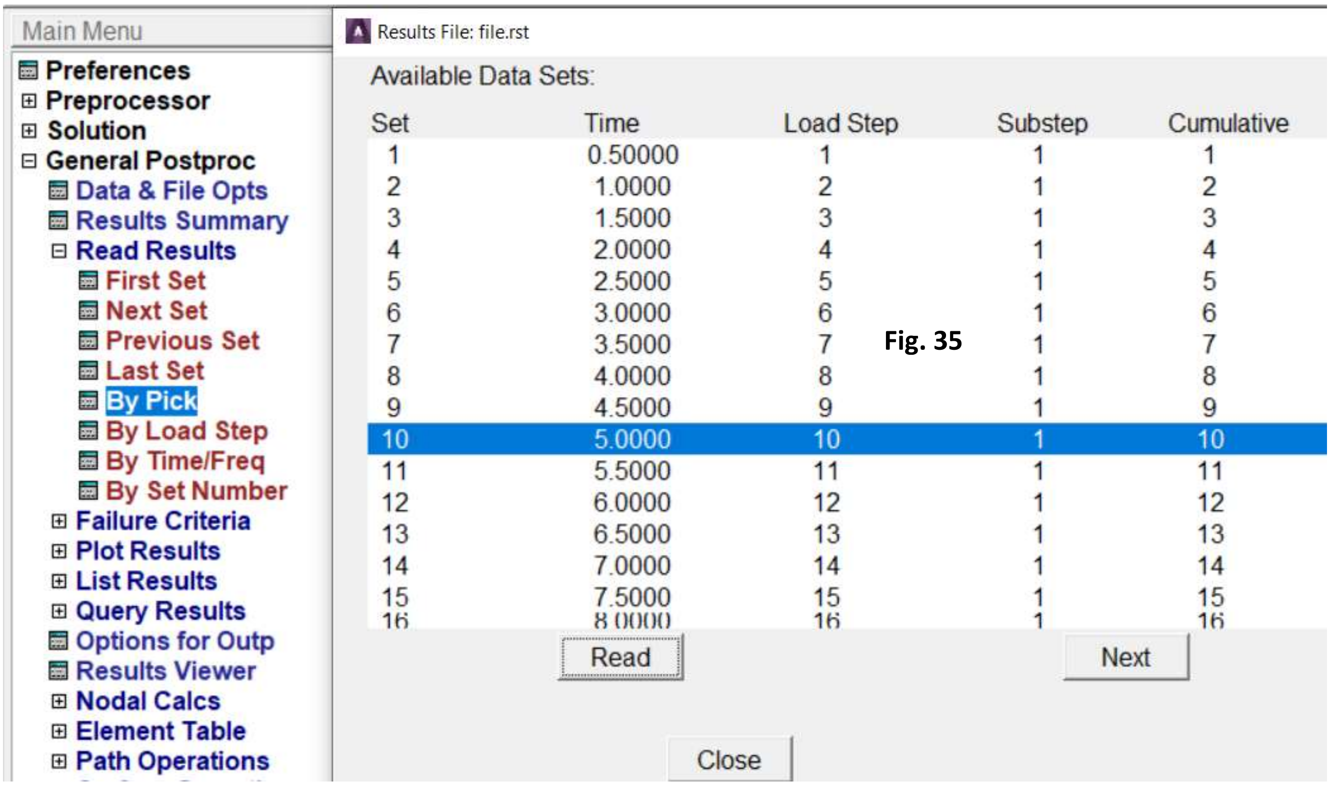


Fig. 35

Discuss results of cases 1-5 and write conclusions.