HEAT TRANSFER PROBLEMS AND THERMAL STRESSES

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(\lambda_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\lambda_y \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(\lambda_z \frac{\partial T}{\partial z}\right) + q_v(x, y, z, t),$$

where: $T(x, y, z, t)$ - temperature, q_v – heat generation (W/m^3), ρ – density (kg/m^3),
$\lambda_x, \lambda_y, \lambda_z$ – thermal conductivity (W/mK), c – specific heat (J/kg).

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most materials do vary with temperature, so the analysis usually is nonlinear.

Transient thermal analysis determines temperatures and other thermal quantities that vary over time. Many heat transfer applications--heat treatment problems, nozzles, engine blocks, piping systems, pressure vessels, etc.--involve transient thermal analyses.

Temperature distribution that a transient thermal analysis calculates is often used as input to structural analyses for thermal stress evaluations.
A transient thermal analysis follows basically the same procedures as a steady-state thermal analysis. The main difference is that most applied loads in a transient analysis are functions of time. To specify time-dependent loads the load-versus-time curve should be divided into load steps.
If you use individual load steps, each "corner" on the load-time curve can be one load step, as shown in the following sketches.
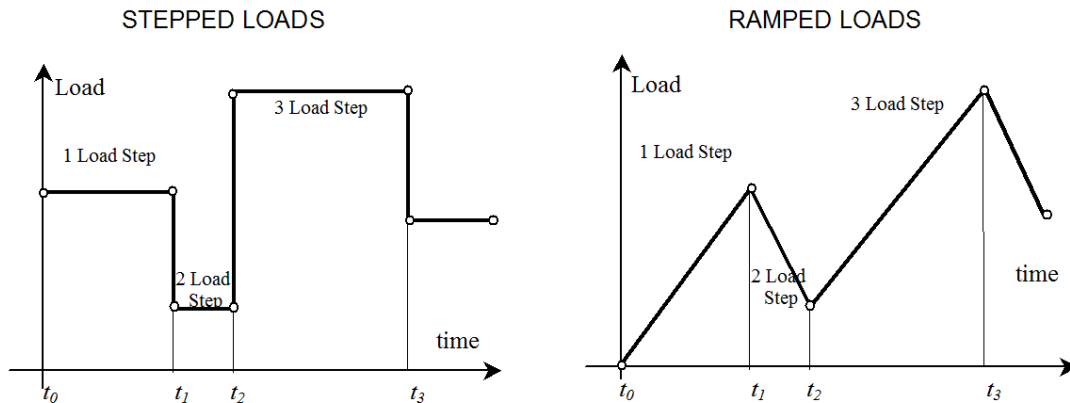
Examples of load-versus-time curves – stepped and ramped loads

For each load step, you need to specify both load values and time values, along with other load step options such as stepped or ramped loads , automatic time stepping, etc. You then write each load step to a file and solve all load steps together.

Temperature-dependent coefficient of thermal expansion α_t **(T)**

If α_t is the thermal expansion coefficient then the typical component of thermal strain is
$\varepsilon_{th} = \alpha_t$ (T) (T- T_0).
If $T_0 = T_{ref}$, where T_{ref} is the reference temperature at which zero strains exist such a coefficient is correctly used. If this condition is not true an adjustment must be made (MPAMOD command in the Preprocessor).

Thermal stresses

Thermal stresses usually are analysed using *sequential method*.

The *sequential method* involves two or more sequential analyses, each belonging to a different field. You couple the two fields by applying *results* from the first analysis as *loads* for the second analysis. In the case of thermal-stress analysis the nodal temperatures from the thermal analysis are applied as "body " loads in the subsequent stress analysis

Ansys enables also using *direct method* which involves just one analysis that uses a *coupled-field element type* containing all necessary degrees of freedom.
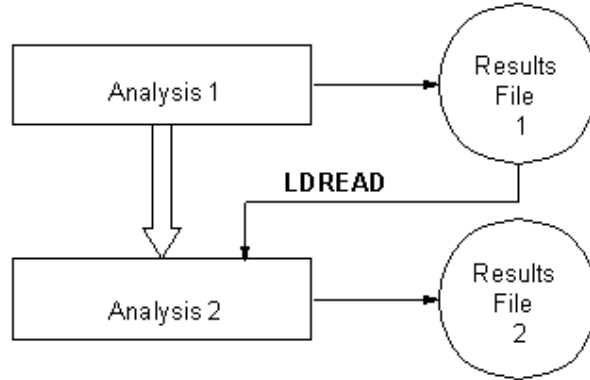
Data flow for a sequential coupled field analysis.

Example 1 - thermal stresses in steady state heat flow

In the steel thick pipe we have the internal temperature T_w=100°C end external temperature T_z=20°C. The inner radius is a=30mm, and outer b=40mm. Show the temperature distribution, von Mises stress and stress components in the cylidrical coordinate system. E=2e11Pa, ν=0.3, $α_t$=1.2e-5 1/K, k=50 W/mK. Consider the pipe constrained in the axial direction at both ends.
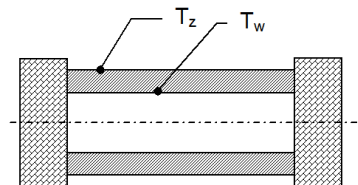
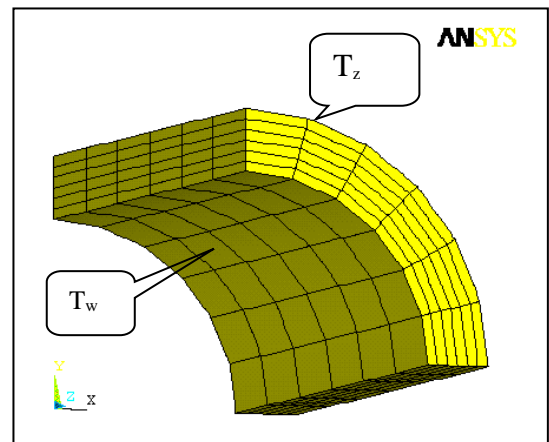Analytical solution (plane strain problem, $\mathbf{T_{ref}=0°C}$):

$$T(r) = T_w + \frac{T_z - T_w}{\ln\left(\frac{b}{a}\right)} \ln\left(\frac{r}{a}\right)$$

$$\sigma_r(r) = C\left[\ln\left(\frac{b}{r}\right) \middle/ \ln\left(\frac{b}{a}\right) - \left(\frac{b^2}{r^2}-1\right) \middle/ \left(\frac{b^2}{a^2}-1\right) \right]$$

$$\sigma_t(r) = C\left[\left(\ln\left(\frac{b}{r}\right)-1\right) \middle/ \ln\left(\frac{b}{a}\right) + \left(\frac{b^2}{r^2}+1\right) \middle/ \left(\frac{b^2}{a^2}-1\right) \right]$$
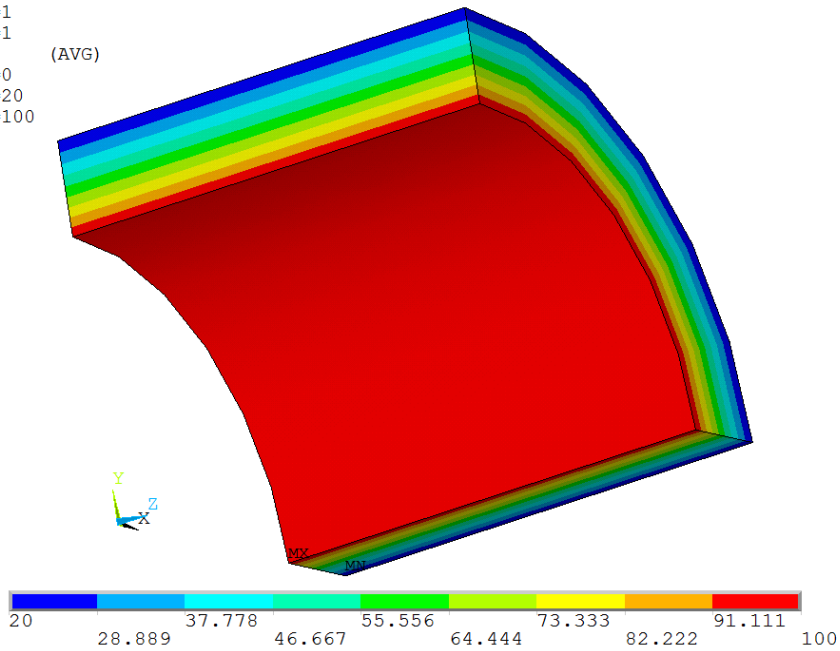
$$\sigma_z(r) = v(\sigma_r + \sigma_t) - \lambda E T$$
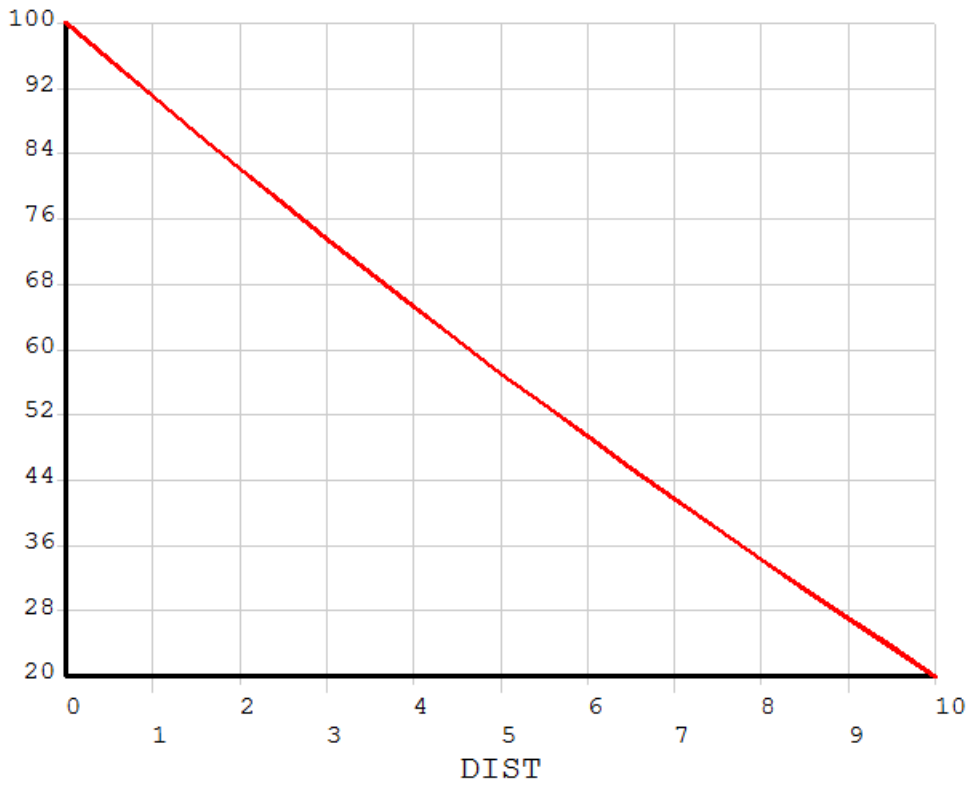
$$C = \frac{-E\alpha_t(T_w - T_z)}{2(1-v)}$$

From the above formulas we have e. g. σ_t (a)=-150.2 MPa, σ_t (b)=124.1 MPa.

Distribution of temperature in the pipe
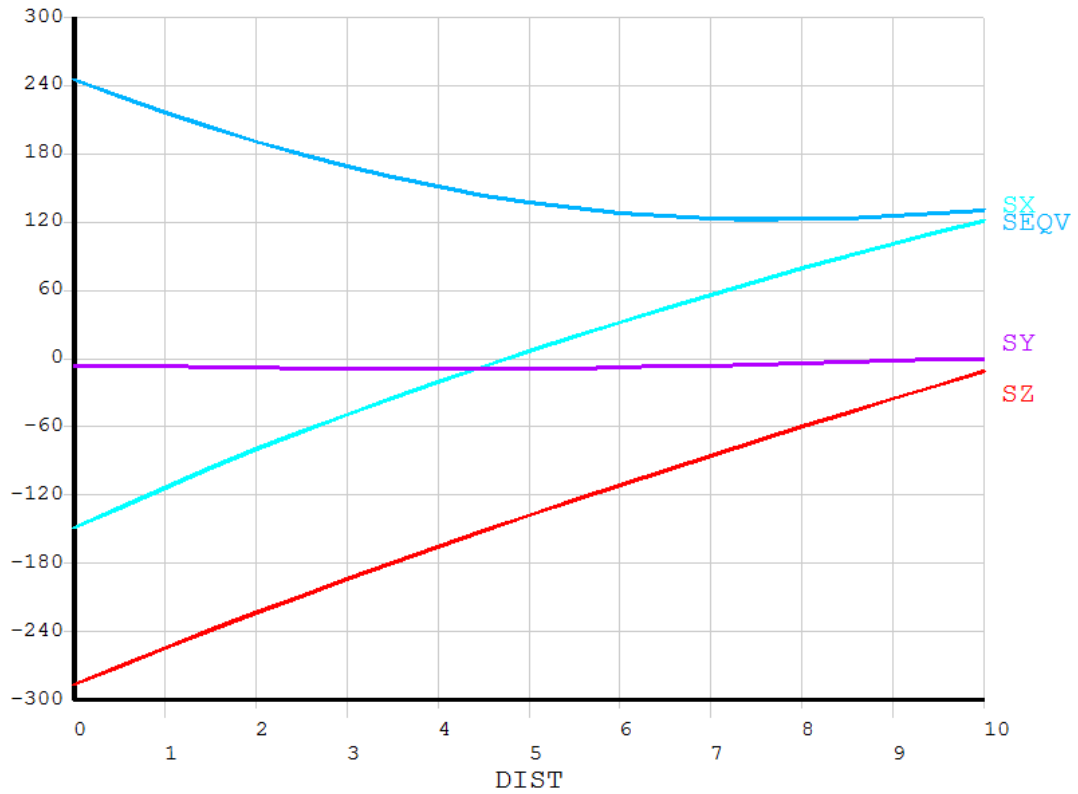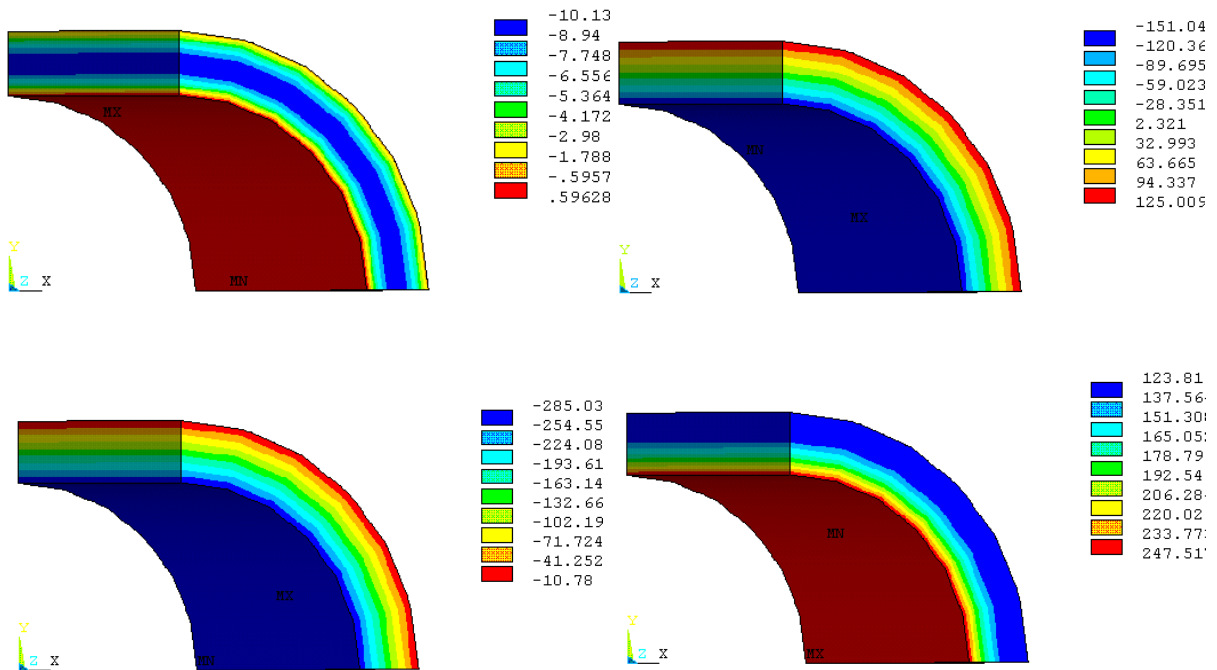

Temperure along the thickness of the pipe

Radial stress (Sx) ,hoop stress (Sy) , axial stress (Sz) and von Mises stress (Sred) along the thickness of the pipe (MPa)

Distribution of radial stress (Sx) ,hoop stress (Sy) , axial stress (Sz)   and von Mises stress (Sred)
Cylindrical coordinate system

Approach

The analysis may be performed using plane strain model (cross-section of the pipe), axisymmetric model, or 3D model. In each case only a segment of the pipe may be analysed with the adequate symmetry conditions.

Summary of steps in numerical analysis (3D):

Preprocessor

-define geometry of the analysed region (part of the cylinder),
-define the material properties (E,ν,λ,k)
-define the element type (*thermal solid*)
-specify the boundary conditions (temperatures T_w and T_z applied on the adequate areas)
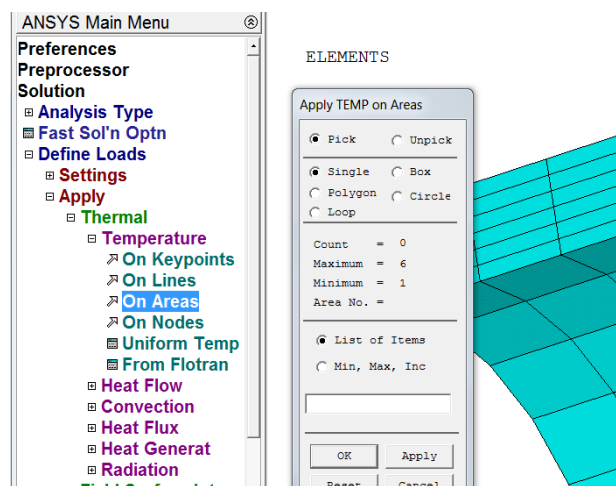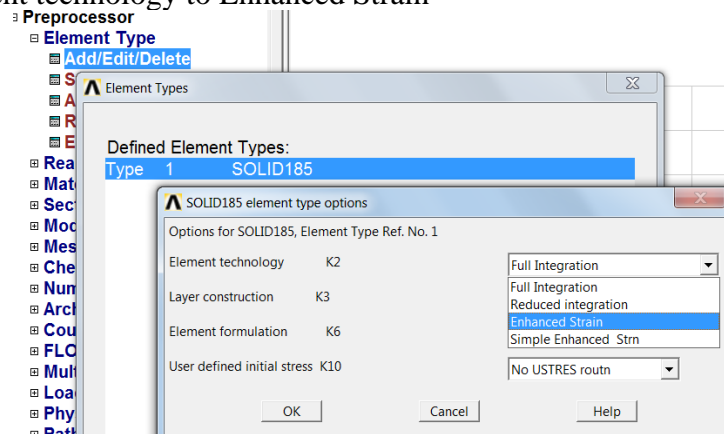

-mesh the model

Solution
-solve the current load step *(Solve>Current Load Step)*

General Postprocessor
-review the results- plot the maps of interest and the temperature as the graph along the path (thickness).

Preprocessor
- change the element type from *thermal solid* to *structural solid* (**Element Type>Switch Element Type***)*
- switch element technology to Enhanced Strain

<u>Solution</u>

    - apply the boundary conditions for stress analysis (*Define Loads>Displacements> Symmetry B.C.>On Areas)*

    - apply the nodal tempertatures as a load in stress analysis (file *jobname.rth* from thermal analysis)
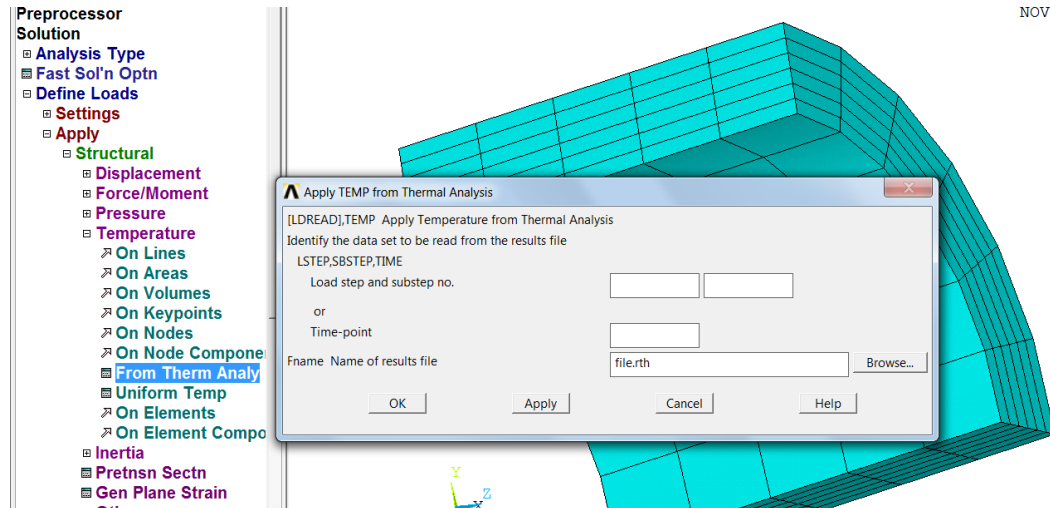
    - solve the current load step

<u>General Postprocessor</u>

    - review the results - plot the stresses as graphs along the path (thickness).
Use cylindrical coordinate system: (*Options For Output>Results Coordinate System*)

Tasks and questions:

    1. Repeat the analysis using 2D plane strain or axisymmetric model. Compare the obtained results with the results corresponding to the 3D model.

    2. Perform the adequate analysis for the pipe with unconstrained ends (without the axial compression). Explain the differences.

    3. Find the results corresponding to point 2, but with the 10mm insulation
(E=1*10^3 MPa, v=0.35, k= 0.1 W/m^{2}K)

    3. Repeat the calculations using the model of convection b.c. :
 At the int. surface : bulk temp. 100C, film coefficient   h=500W/(m^{2}K)
At the ext. surface: bulk temperature 20C, film coefficient is   h=10W/(m^{2}K).
Explain the results.

In each case save the results: FE mesh, temperature distribution and stress components distributions in the form of contour plots and graphs along the path (thickness).

Example 2 - thermal stresses in transient heat flow

Steel balls of diameter d=12mm are heated to T_1=850°C and then quenched in oil. The temperature of the oil is assumed as constant T_o=40°C. Heat exchange coefficient at the surface oil – the ball is   h=400W/(m²K).

How long should the balls stay within the oil bath to get the temperature T_2= 100°C in the centre?

What is the maximum von Mises stress during the process?

Material properties of the steel :   density ρ=7800kg/m³, specific heat c=444J/(kgK), thermal conductivity k=40W/mK, coefficient of thermal expansion $α_t$=1.2e-5 1/K, Young modulus E=2e11Pa, Poisson's ratio ν=0.3.
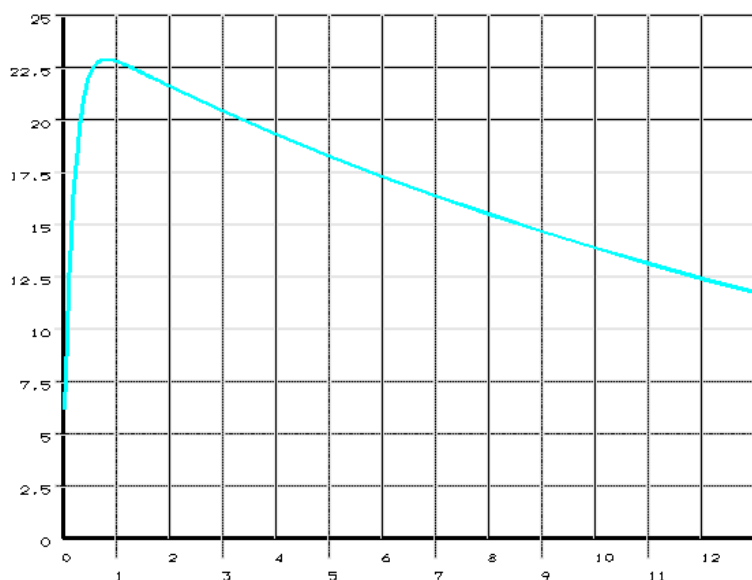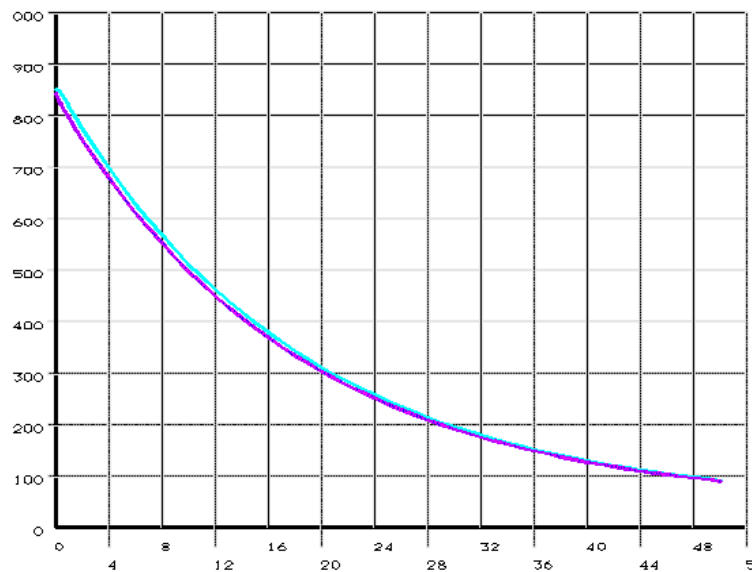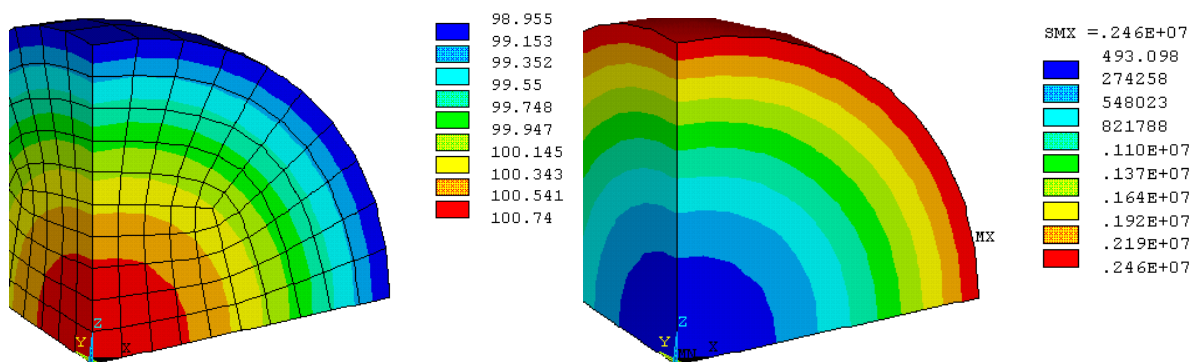
Temperature of the ball as the function of time
a)   temperature in the centreTw and at the surface (Tz)
b)   the difference DT=Tw-Tz
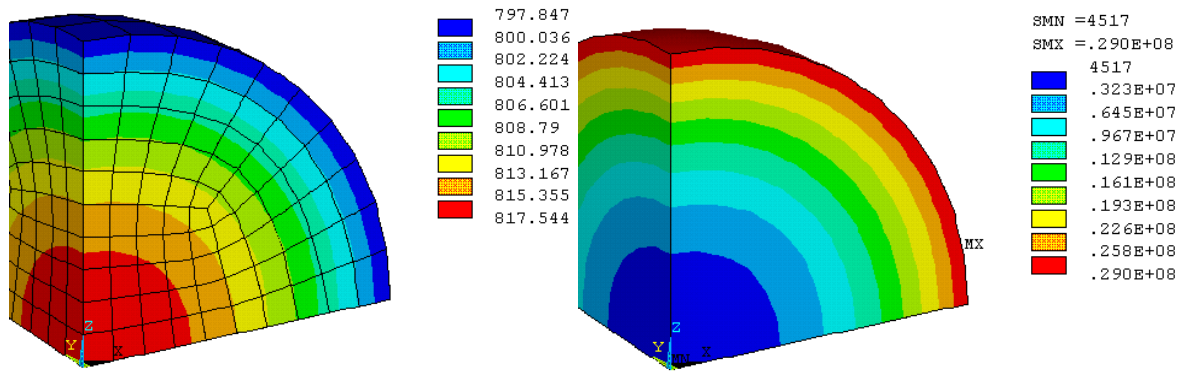
Temperature distribution (°C) and von Mises stress (Pa) after 1 s of cooling

Temperature distribution (°C) and von Mises stress (Pa) after 48 seconds of cooling.

Approach

The analysis may be performed using axisymmetric or 3D model. In each case only a segment of the ball may be analysed with the adequate symmetry conditions.

## Summary of steps in numerical analysis (3D):

Preprocessor
        -define geometry of analysed region (part of the sphere by intersection of sphere and block),
        -define material properties (E,ν,λ,k, ρ,c)
        -define element type (*thermal solid*)
        -specify the boundary conditions (convection on external surface with *bulk temperature* 40°C, *film coefficient* 400W/m^{2}K)
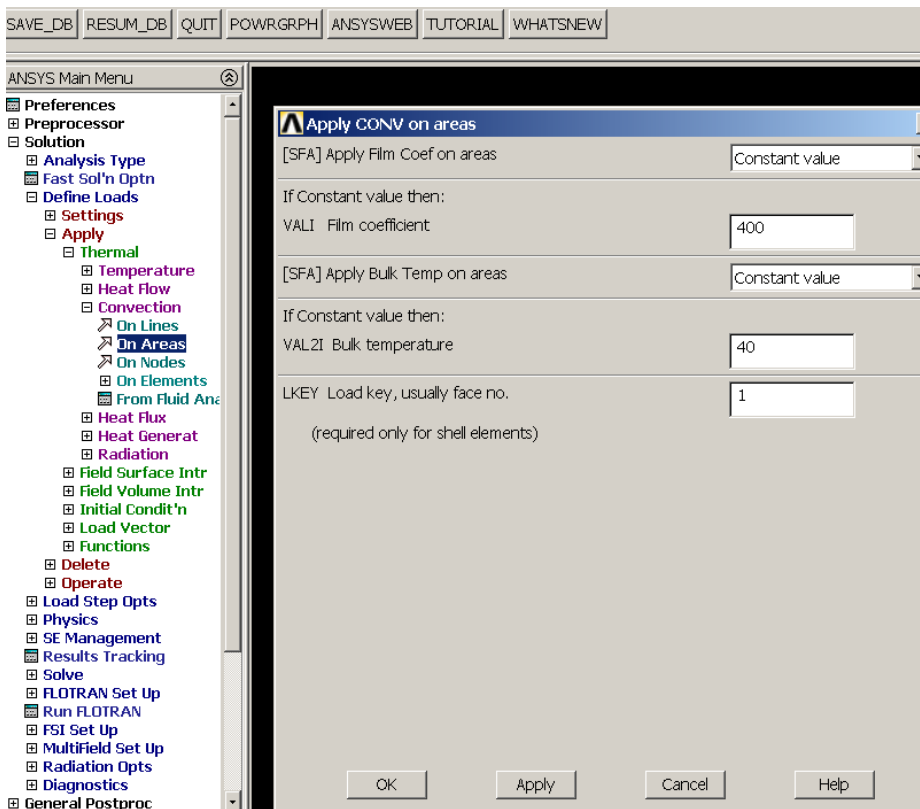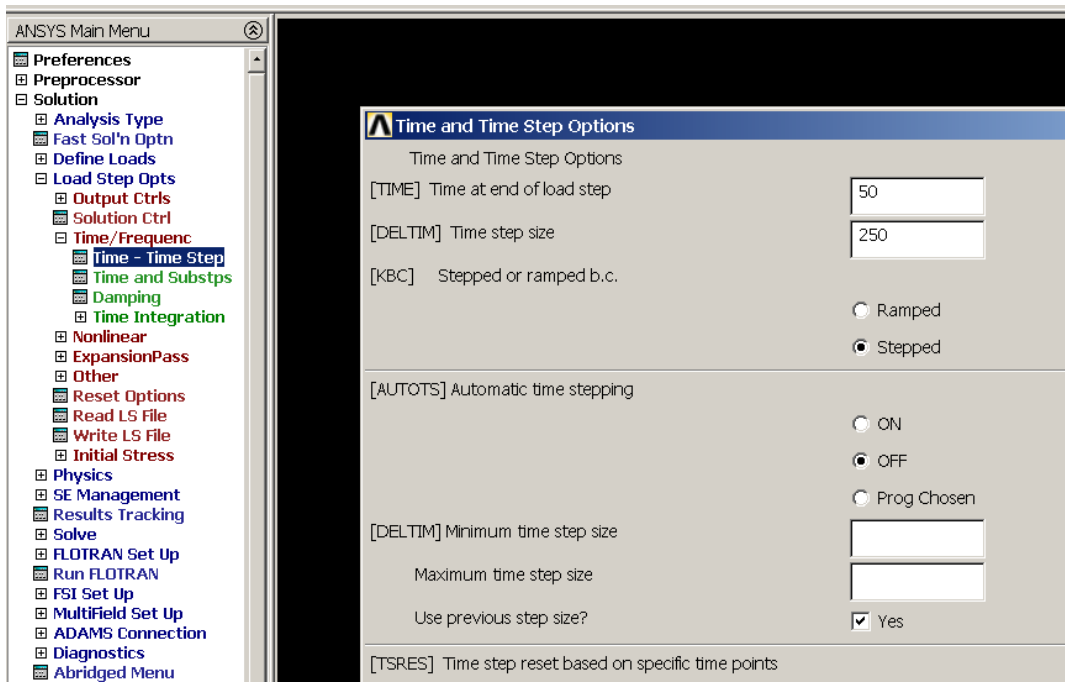
-mesh the model

Solution

- set type of analysis – ANALYSIS TYPE   - *transient* (solution)-
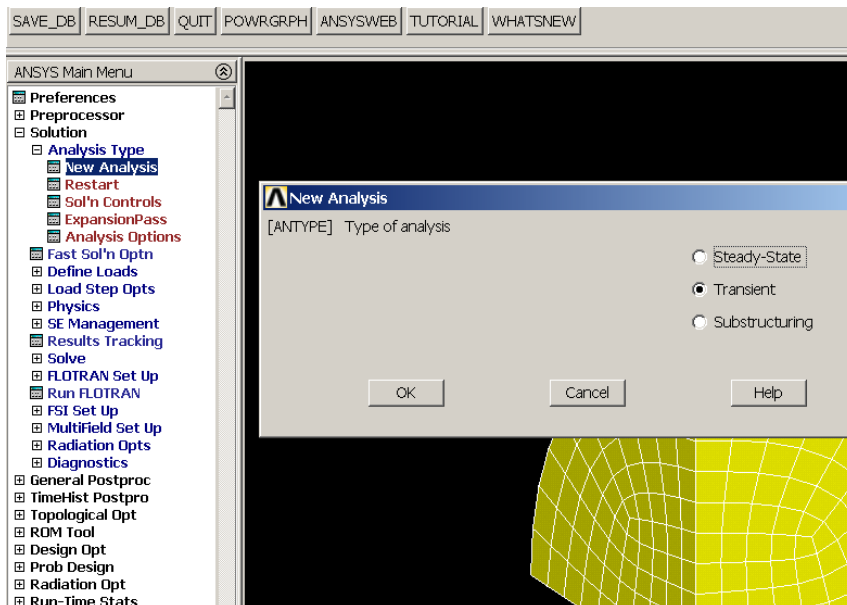
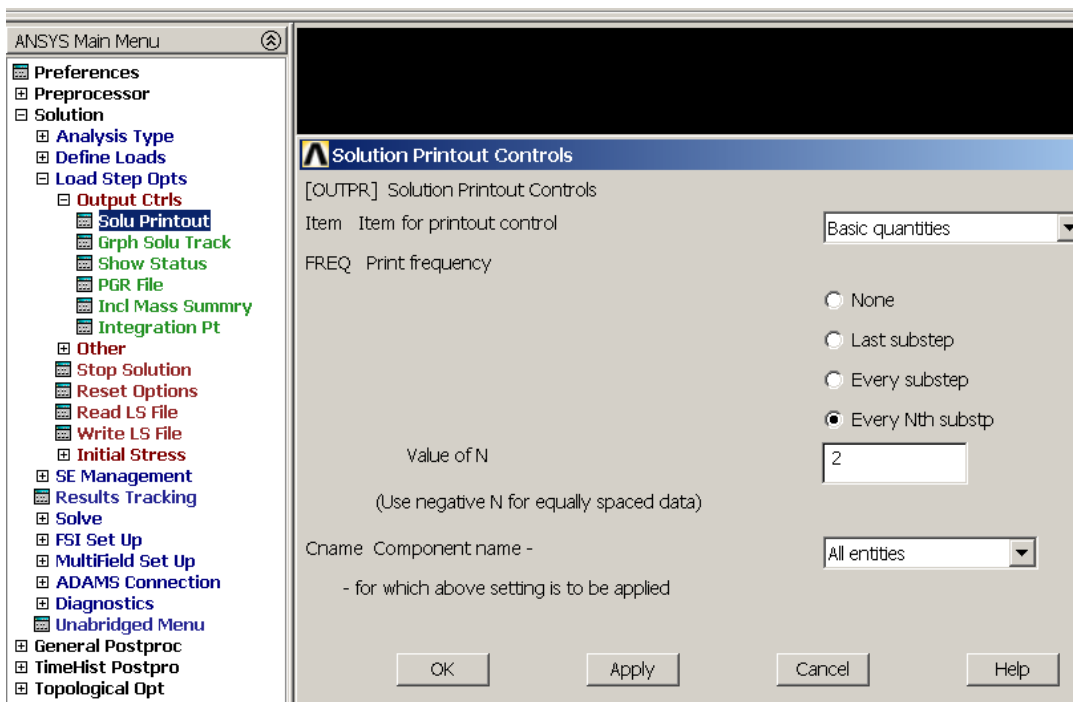- set initial condition TREF= 850°C (being the initial temperature)
-set time, time step and related parametrs (Load step options), e.g.:
              time=50sec, time step 0.2 sec., stepped, auto time step OFF
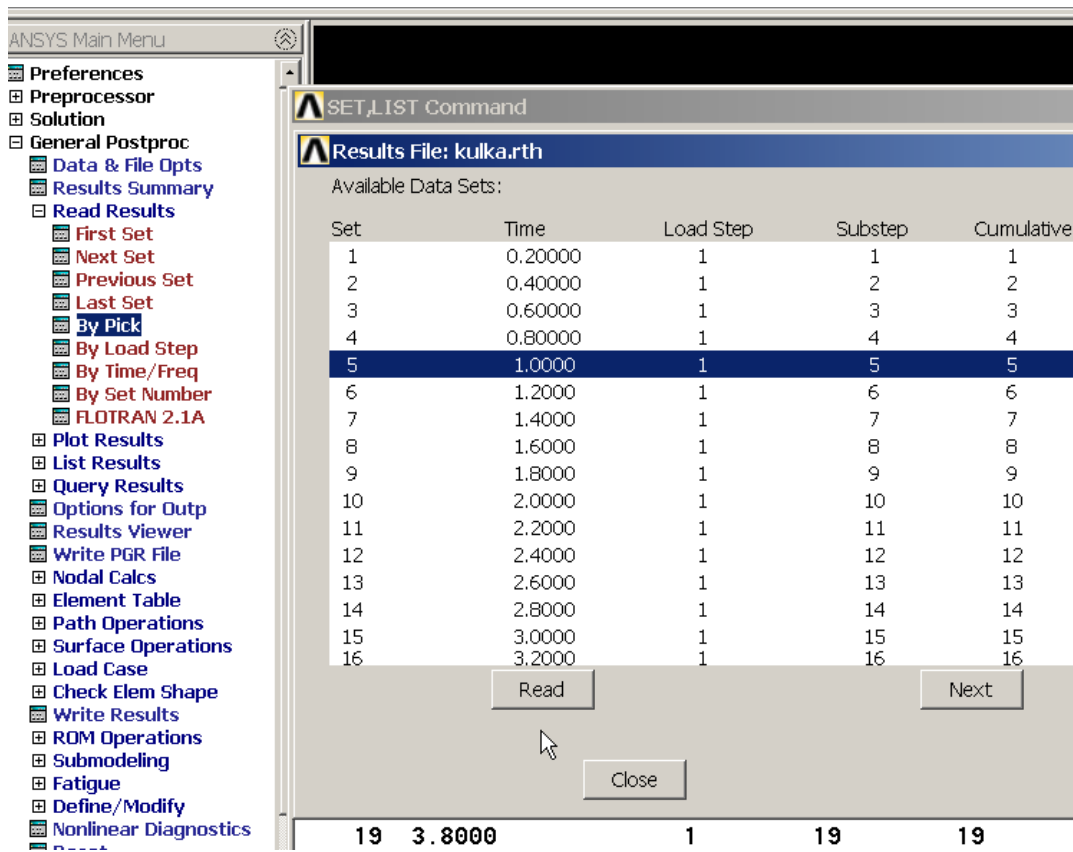
-set

output controls (file write sequence to *every substep or every Nth step*)
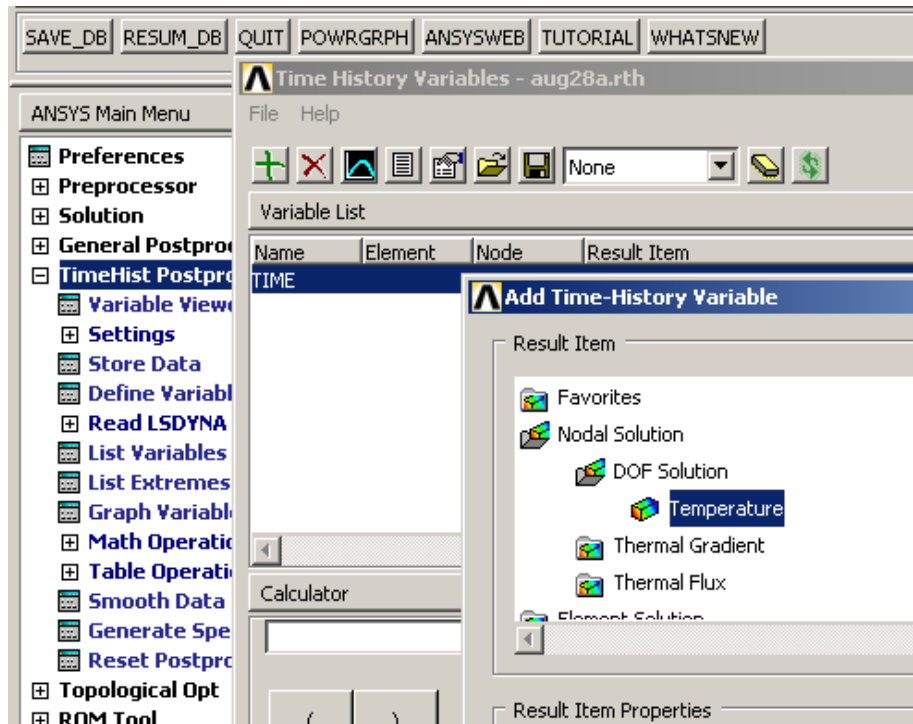

-solve Current Load Step

General Postprocessor

Read the results corresponding to the chosen time point

-review the results corresponding to the arbitrary point of time
-animate the results (General Postprocessor – PlotCTrl)

TimeHist Postprocessor

-review the results e.g. variation of temperature with respect to time at chosen points (commands *Define Variables i Graph Variables)*:
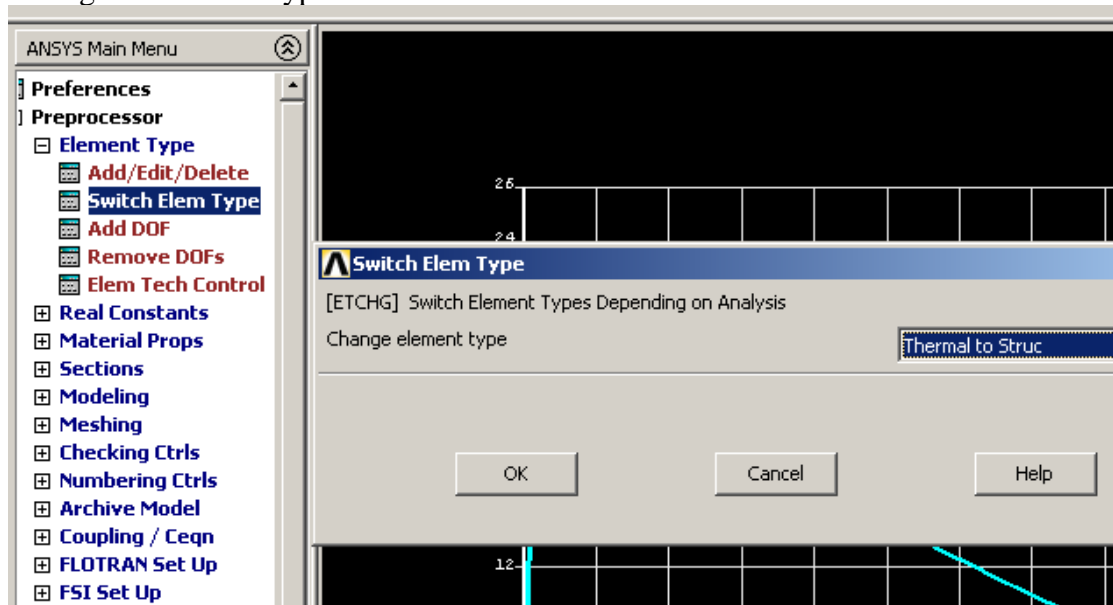

-find the difference between the temperature in the centre of the ball and the surface – delta T*:*
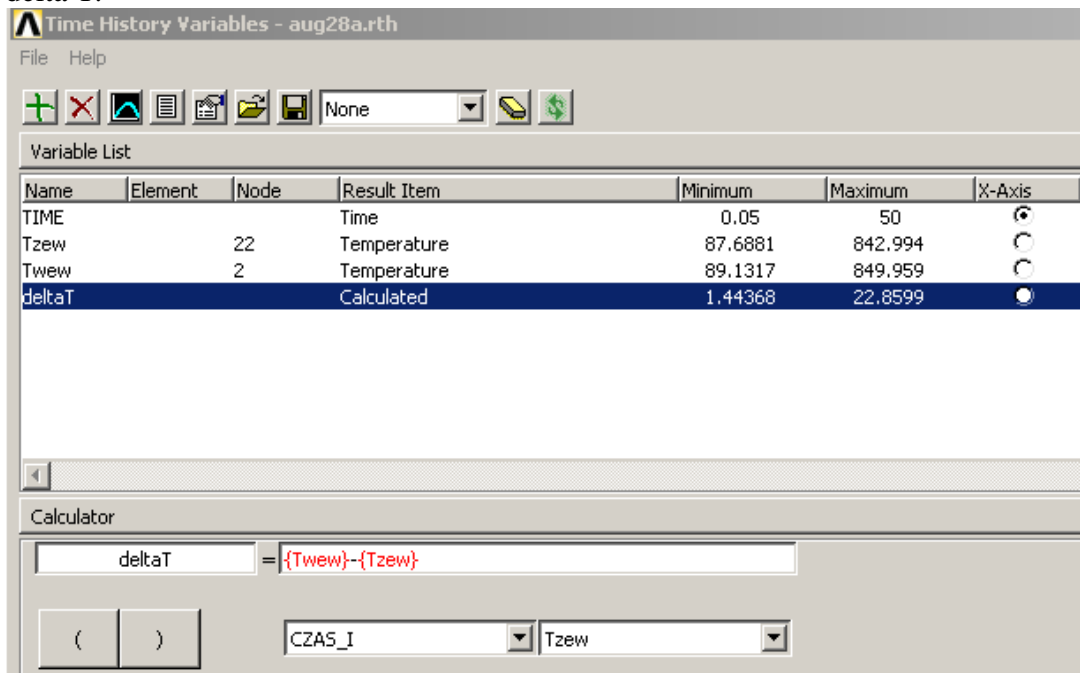
-show the defined functions in the form of graphs

<u>Preprocessor</u>

    -change the element type from *thermal solid* to *structural solid*

-apply the boundary conditions for stress analysis (symmetry)

-apply the nodal tempertatures as a load in stress analysis at the arbitrary point of time t_{arb} (file *jobname.rth*)
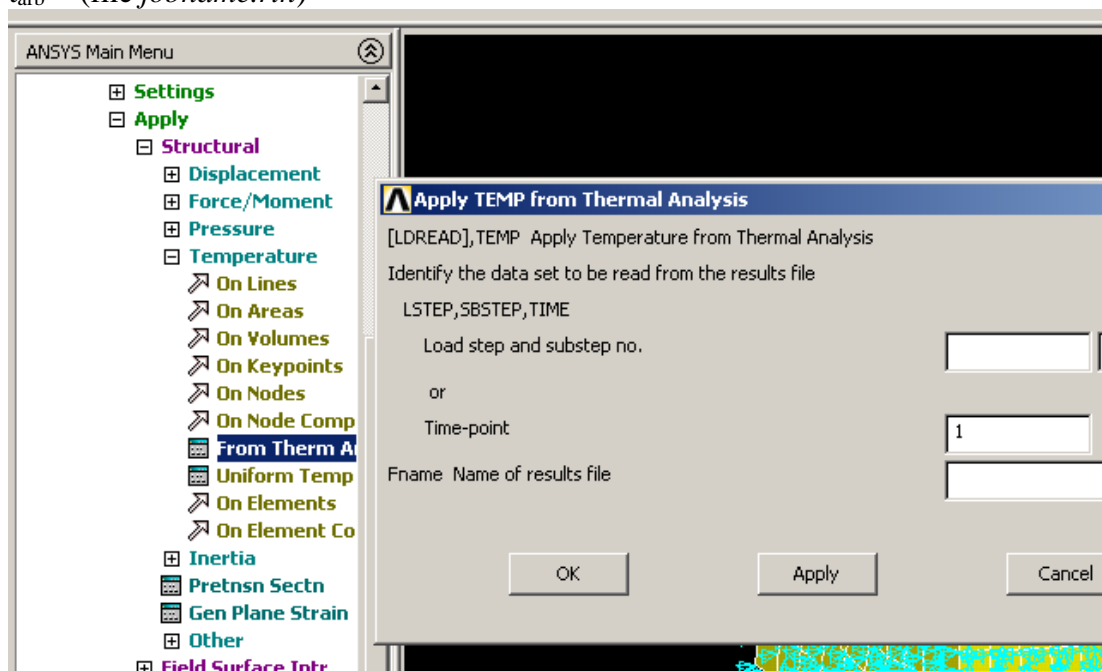
<u>Solution</u>

    - set analysis type to static

    -set time=1 , analysis using 1 substep (Time/Frequency)

    -solve Current Load Step

<u>General Postprocessor</u>

    -Review the results for the time t_{arb}

In the presented way of analysis the thermal stresses corresponding to the chosen time t_{arb} were presented. If we want to find the history of stresses in a period of time the corresponding history of loads for structural analysis should be built in the form of Load Step files (Files *Jobname.s i* , where i is the number of the load step*).*

It may be also performed automatically using for example the presented below macro, which is set of commands written using Ansys Parametric Design Lenguage (APDL).
The file may be read by the program in Solution phase using the INPUT command.

```
/COM, **************************************************
/COM, Creation of files jobname.si for history of linear thermal stresses
/COM, The commands may be input after changing elements to structural and
/COM, after applying the adequate structural boundary conditions
      *ask,case,name of the file rth (jobname) =,'file'
      *ask,time_in,initial time =,1
      *ask,time_e,final time  =,1
      *ask,liczba_p,number of time segments = ,1
      przyr=(time_e-time_in)/liczba_p
      chwila=time_in

      ANTYPE,0
      *DO,k,1,liczba_p
      LDREAD,TEMP,,,chwila, ,case,'rth',
      TIME,chwila
      AUTOTS,0
      NSUBST,1, , ,1
      KBC,1
      LSWRITE,k,
      chwila=chwila+przyr
      *ENDDO
```

Tasks and questions:

-Repeat the analysis using axisymmetric model. Compare the results.

-Find the influence of heat exchange coefficient and thermal conductivity on the results (maximum stresses and time of cooling)

-Perform the analysis using the elasto-plastic model of the material. Why in this case residual stresses may be expected?