Exercise 4 (v.2023a)
Temperature distribution and thermal stresses

Introduction

The finite element method allows you to effectively calculate thermal stresses. Thermal strains due to thermal expansion in a three-dimensional, isotropic body are described by the strain vector:

$$\boldsymbol{\epsilon}\,_T^T = \begin{bmatrix} \epsilon_{xT} & \epsilon_{yT} & \epsilon_{zT} & \gamma_{xyT} & \gamma_{yzT} & \gamma_{xzT} \end{bmatrix} = [\alpha\Delta T\ \alpha\Delta T\ \alpha\Delta T\ 0\ 0\ 0]$$

The total strains are then the sum of thermal strains and elastic strains (caused by a stress field):

$$\boldsymbol{\epsilon} = \boldsymbol{\epsilon}\,_T + \boldsymbol{\epsilon}\,_S$$

The problem of thermal stresses is usually solved in two steps:
1. Solution for resultant temperature in a stationary problem (*steady state*) or in a transient *heat* flow. The results are saved in the *jobname.rth file*
2. Solving the problem of stress analysis including a thermal load – i.e. taking into account additional nodal forces resulting from temperature changes. These forces are calculated basing on information about nodal temperatures from the *jobname.rth file*

The heat conduction equation has the form:

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x_i}\left(\lambda_{x_i}\frac{\partial T}{\partial x_i}\right) + q_V$$

In the finite element method, instead of a PDE, an equivalent formulation of the minimization of the functional is used, which after discretization is transformed into a system of algebraic equations. The finite elements used in the calculation of temperature fields are corresponding to the finite elements constructed for stress analysis and have analogous shapes, numbers of nodes and shape functions. At each node, we only have one degree of freedom. Usually, solving the conduction equation using FEM is not a problem.

**boundary conditions** that may occur in the calculations :
1. temperature
2. heat flux density (W/m^2)
3. heat flux (nodal force)
4. Newton's law of convection: $q = \alpha_k(T_b - T_S)$
5. radiation

The properties of a medium may be functions of temperature. The boundary conditions can be variable in time, applied in shocks (*Step- Loads*) or gradually (*Ramped-Loads*). In the case of complex processes, the course of changes in boundary conditions is presented as a sequence of subsequent loading steps (*Load Steps*). Calculations of transient temperature fields also require initial conditions.

The first case – temperature and stress <u>in the steady state</u>

	Material :
Thick-wall pipe – thermal load	steel E = 200 GPa
inner radius: a = 30 mm	ni = 0.3
external radius: b = 40 mm (*length is arbitrary, e.g. = 60 mm*)	alpha = 1.2e-5 1/K
	lambda = 50 W/(m K)
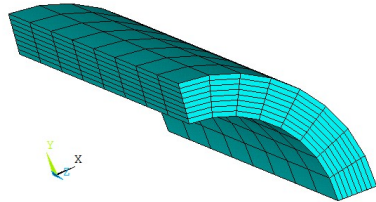
internal surface temperature: $T_w = 100^o C$
external surface temperature: $T_z = 20^o C$

The temperature distribution should be calculated, as well as the stress distributions: circumferential σ_t, radial σ_r, axial σ_z and equivalent stress HM. Assume initially that the pipe cross-sections remain in a plane state of deformation.
The analytical solution shows that $\sigma_t(a) = -150.2 MPa$, $\sigma_t(b) = 124.1 MPa$.
FEM calculations will be performed for the option: three-dimensional pipe section.

NOTE on the choice of set of units: SI *(N, m, s, W, kg)* **or mod_SI** *(N, mm, s, mW , tons)* **.**

Temperature distribution calculation:

Preprocessor :
0. Preferences – **(set <u>BOTH</u> types of analyses: *Structural* and *Thermal*))**
1. Geomety
   *Modeling > Create > Volumes > Cylinder > By Dimensions (**cylinder length 60 mm = 0.06 m, angle 90 deg**)*
2. Material properties: E, nu , lambda, alpha **(NOTE the units: SI or mod_SI)**
   Material Properties > Material Models > Structural > Linear > Elastic > Isotropic
   Material Properties > Material Models > Structural > Thermal Ex. > Secant Coeff. > Isotropic
   Material Properties > Material Models > Thermal > Conductivity > Isotropic
3. Element choice
   *Element Type > Add > Thermal Mass > Solid (**select Brick20node - 279**)*

4. Meshing into finite elements **(HEX elements, i.e. Mapped division)**
   *Meshing > Mesh Tools > Lines - Set > Pick All (**division everyone lines for 8 elements**)*
   *Meshing > Mesh Tools > Volumes (**note : select HEX!!**) **Check List>Elem (how many elems ?)***

Solution
5. Specify the type of analysis
   Analysis Type > New Analysis > Steady State
6. Introduction of boundary conditions: temperatures on the <u>*inner*</u> and <u>outer</u> surfaces of the pipe
   Define Loads > Apply > Thermal > Temperature > On Areas
   ***Check** !: List > Loads > Solid Model Loads (how many/ **which surfaces should there be** ??)*
   Area numbering: Plot> Areas and PlotCtrls > Numbering … and Replot
7. Calculation of the temperature distribution inside of solid (thru-thickness plot)
   Solve > Current Load Step

General Postprocessor

8. Assessment of heat flow results and preparation of plots illustrating the solution
   Plot Results > Contour Plot > Nodal Solution > DOF Solution > Nodal temperature

9. Temperature graph along the pipe thickness
   steps for: defining the path, mapping the temperature along
   the path and presenting it in the form of a graph:

a. *PathOperations > Define Path > ByNodes* **(indicate 2 nodes, on internal and external surface and Name the path)**

b. *Map onto Path* **(Name the variable e.g. T_r and select DOF: Temp)**

c. *Plot Path Item > On Graph* **(select T_r)**

All thermal analysis results are written in the ***jobname.rth file***
and will be used to determine thermal loads during stress
analysis.
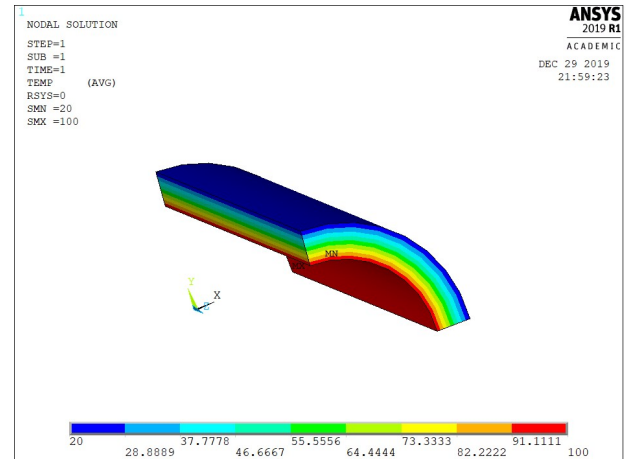
Calculating the stress distribution:

After evaluating the temperature distribution results, we move back to the Preprocessor. Convert the FE to a stress analysis model - i.e. change the element type (with the same meshing)

Preprocessor :

10. Change a *Thermal Solid element* to the corresponding *Structural Solid element*
   *Element Type > Switch Element Type > Thermal to Struc .*

Solution

11. Introduce the displacement boundary conditions on 4 surfaces (four FLAT cross-sections) of the model
    Define Loads > Apply > Structural > Displacements > Symmetry BC > On Areas

12. Enter the loads in the form of nodal temperatures obtained as the result of heat flow analysis (*jobname.rth file*) - as loads in the stress analysis:

    Define Loads > Apply > Structural > Temperature > From Thermal Analysis (plus *Solution Opts – Output Ctrls)*

13. Calculation of the stress and displacement field
    Solve > Current Load Step

General Postprocessor

14. In the case of cylindrical pipe, it is convenient to use RSYS1 to present the results: cylindrical coordinate system (default is: Z axis along the cylinder axis):
    Option For Output > Results Coordinate System (set **Global Cylindrical)**

15. Evaluation of the results in the form of a stress and displacement map and graphs
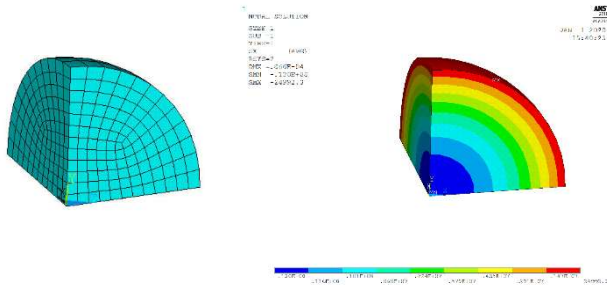   *Plot Results > Contour Plot > Nodal Solution > DOF Solution*

(Sample maps below - in SI units)
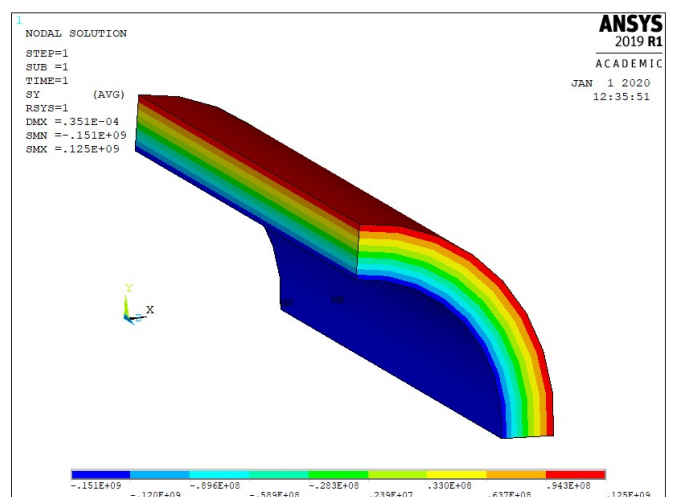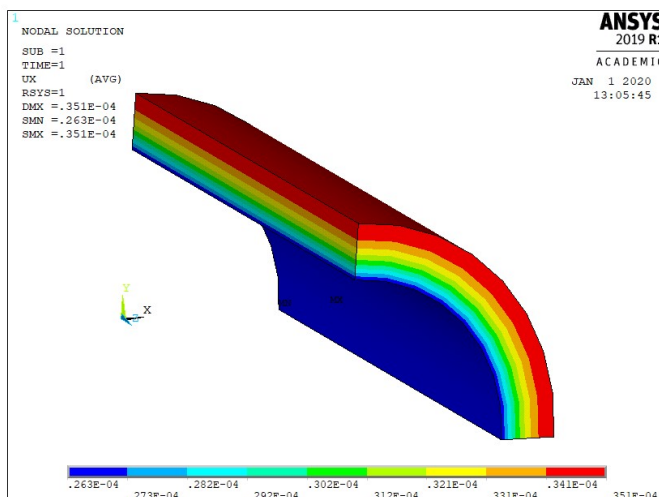
The second case - <u>transient states</u> of temperature and stress

<table>
<tr>
<td>

Hardening of the steel ball

Radius = 6mm   (only 1/8 of the ball is analyzed due to symmetry)
initial temperature of the ball: $T_p = 850^o C$
oil temperature (constant): $T_o = 40^o C$

</td>
<td>

Ball material :

E = 2e5 MPa (<u>change to SI units</u>)
nu = 0.3
density ρ = 7800 kg/m ^{3}
coef. thermal expansion (CTE) λ = 1.2e-5   1/K
coef. heat conduction k = 40 W/(m K)
specific heat c = 444 J/(kg K)
film coeff. (convection heat transf) $\alpha = 400\ W/(m^2 K)$

</td>
</tr>
</table>

- How long should the ball stay in the oil bath to reach a temperature of 100 degrees inside?
- When will the maximum equivalent stress in the ball occur?

 The most interesting parameter in the example is*: the temperature variation on the surface and inside the ball*

NOTE on the choice of units : SI *(N, m, s, W, kg)* or mod_SI *(N, mm, s, mW , t).*

Temperature distribution calculation:

Preprocessor :
   0. Preferences – **(set <u>BOTH</u> analyses: *Structural* and *Thermal*))**
   1. Geometry
      (we create object through the intersection of the sphere and the hex-block)
      Modeling > Create > Volumes > Sphere > By Dimensions
      Modeling > Create > Volumes > Block > By Dimensions
      Modeling > Operate > Booleans > Intersect > Common > Volumes
   2. Determination of material properties: E, nu , rho , lambda, alpha
      Material Properties > Material Models > Structural > Linear > Elastic > Isotropic
      Material Properties > Material Models > Structural > Density
      Material Properties > Material Models > Structural > Thermal Ex. > Secant Coeff. >Isotropic
      Material Properties > Material Models > Thermal > Conductivity > Isotropic
      Material Properties > Material Models > Thermal > Specific Heat   *(different value for mod_SI !!!)*
   3. Element choice
      Element Type > Add > Thermal Mass > Solid 279

   4. Meshing into finite elements – all lines into 12 elements *(HEX, Mapped mesh)*
      Meshing > Mesh Tools > Lines - Set > Pick All > No of element divisions = 12
      Meshing > Mesh Tools > Volumes
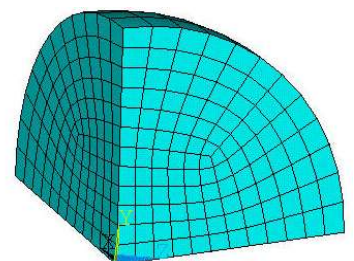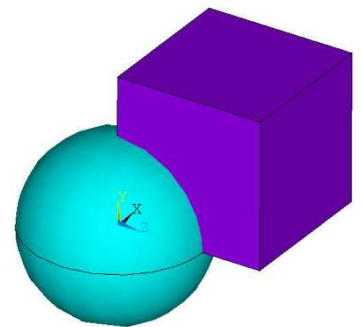
Solution
   5. Specifying the type of analysis
         Analysis Type > New Analysis > Transient
   6. Introduce the THERMAL boundary conditions at the external surface of the analyzed object (convective heat exchange on the <u>OUTER</u> surface)
         Define Loads > Apply > Thermal > Convection > On Areas (... Select Area.) *> Film Coeff. = 400 and BulkTemp =40*
                                       (numeric values - different for mod   SI units!!! 0.4)
   7. Enter the reference temperature and initial temperature
         Define Loads > Settings > Uniform temp. (850 C)
   *Define Loads > Settings > Reference temp. (40 C)*

8. Set the transient analysis details for the job
   (final time = 60 s, number of substeps = 300, **Stepped** impact load , results written for each substep)
      Load Step Opts > OutputCtrls > DB/Results File > Every substep
      Load Step Opts > Time/ Frequenc > Time and Substeps > end = 60; substeps = 300, Stepped; automatic time stepping = off
9. Calculation of the temperature field
      Solve > Current Load Step

General Postprocessor
10. For a selected substep (or better - **for chosen time points**) – evaluate the results,
      Read Results > By Pick

TimeHist Postproc

In this post-processor you can list or graph the results as functions of time. In the drawing on the right: a graph of the temperature **difference** between the node at the surface and in the center of the sphere.
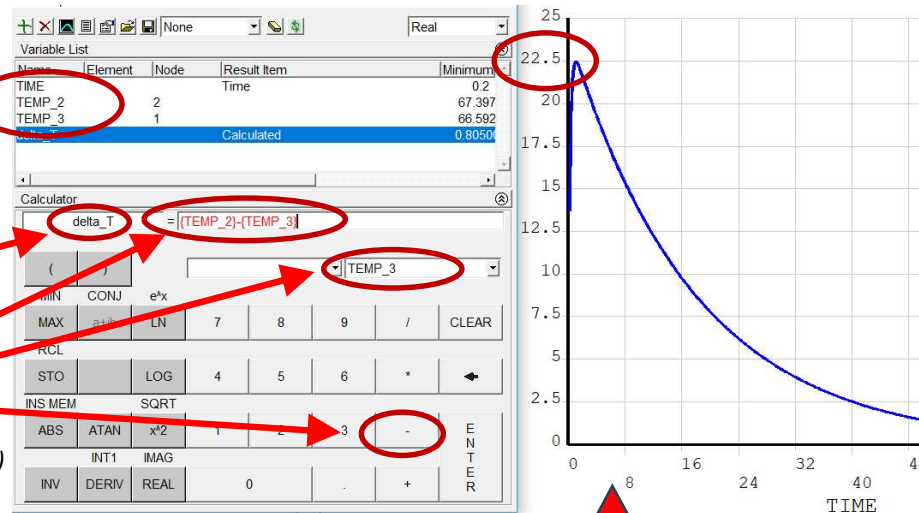
a. Name the values in nodes (plus sign icon)
b. Name for the math operation
c. Create an expression - **difference of variables** by inserting *variables from the drop-down field* and using the calculator
d. Hit **ENTER** on the calculator
e. Create a chart (the third icon - after the plus sign)

NOTE:
the max value on the graph should be ca. **22.5** deg)

Calculating the stress state:
The procedure is the same as in the case of a thick-walled pipe (see points 10-15 for the thick-walled pipe):

Preprocessor:
11. Change a *Thermal Solid element* to the corresponding *Structural Solid element*
      Element Type > Switch Element Type > Thermal to Struc

Solution
12. Set the symmetry displacement boundary conditions on the cross-sectional (flat) surfaces of the model
      Define Loads > Apply > Structural > Displacements > Symmetry BC > On Areas

13. Enter the thermal loads - nodal temperatures obtained as a result of heat flow analysis (*jobname.rth file*) as loads in the stress analysis. Enter **the time** for which thermal loads are to be read (*as time , e.g 48 sec*):

      Define Loads > Apply > Structural > Temperature > From Thermal Analysis

14. Change the analysis type to **Static** and delete substeps (see point 8: **Time over Substeps** : e.g 48 and 1)
15. Calculate the stress and displacement field
   *Solve > Current Load Step*

General Postprocessor
16. Evaluate the results in the form of a stress and displacement map and graphs
      a. *Option For Output > Results Coordinate System > spherical*
        (you may need to load current results: *Read Results >Last step*)
      b. *Plot Results > Contour Plot > Nodal Solution > DOF Solution*

Sample results for typical moments (old results ??):
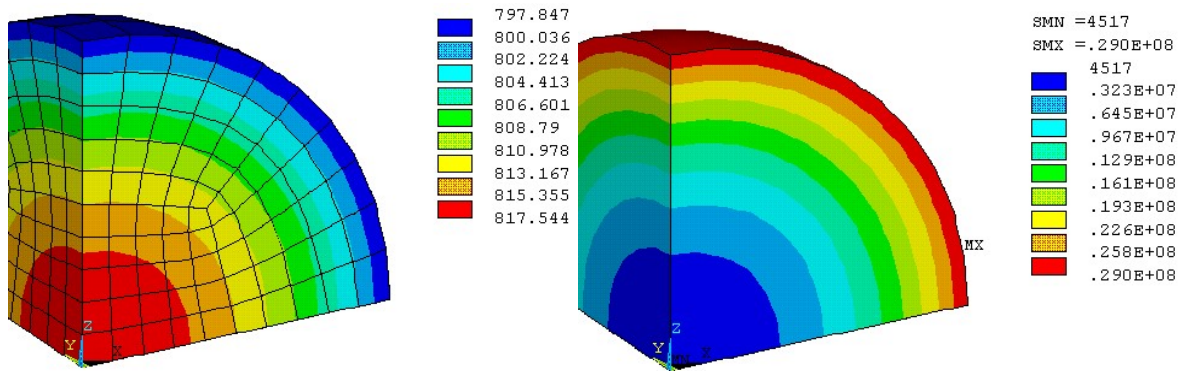
Temperature distribution (°C) and von Mises stress (Pa) after 1 s (???) of cooling
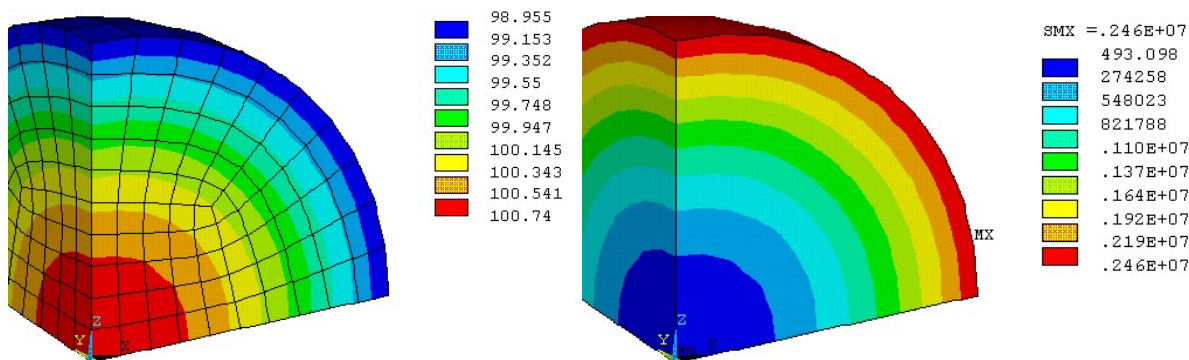
Temperature distribution (°C) and von Mises stress (Pa) after 48 (???) seconds of cooling.

Discussion of the results for the report:

1. The first Case:
Distribution of temperatures, displacements and stresses for **a THICK-WALL PIPE** (*Map of temperatures, displacements, radial, circumferential, axial and reduced stresses)*

2. The second Case:
Distribution of displacement, temperature and equivalent stresse for **a HARDENED BALL** **for time steps** (t=1 sec , t=48 sec)

And of course, Interpretation of the results

Drawings must be on **a WHITE** background (e.g. *Plot Controls / Style / Colors / Reverse Video*)