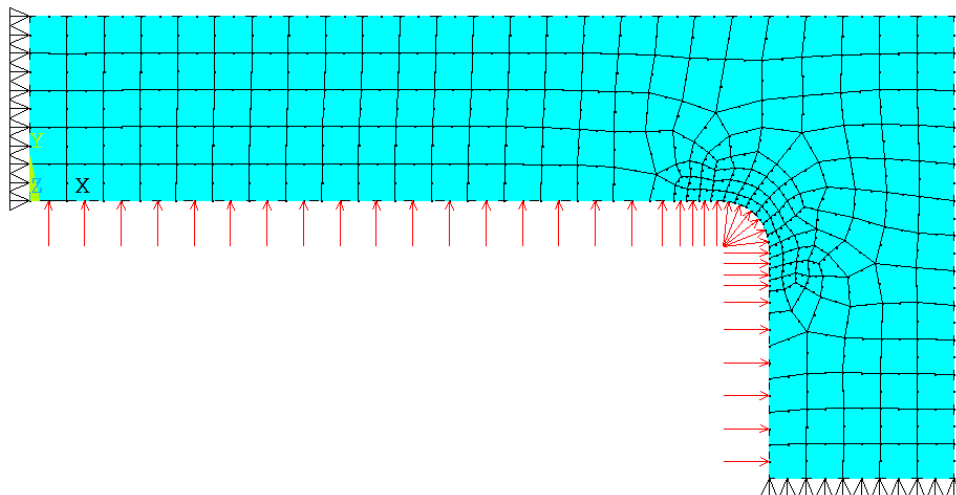# CASE STUDY
ANSYS THERMAL ANALYSIS EXAMPLE

A steel conduit with rectangular cross section transports water at a temperature of 200°F and a pressure of 500 psi. The forced convection coefficient for the running water is $h_{water}=5.80\times10^{-3}$ BTU/sec/in^2/°F. The pipe has a cooling system on the outside that, through a fan, blows air at a temperature of 0°F. The forced convection coefficient for the air is $h_{air}=1.93\times10^{-4}$ BTU/sec/in^2/°F. The material properties for the steel are: $E=30\times10^6$ psi, $\nu=0.29$, K=thermal conductivity=4.63×10^{-4} BTU/sec/in/°F (KXX in ANSYS), and α=coefficient of thermal expansion=8.0×10^{-6} in/in/°F (ALPX in ANSYS). The strain-free temperature for the conduit is 75°F (temperature at which no thermal strains exist in the model).

Using 8-noded quadrilateral elements in ANSYS (PLANE183) determine the von Mises stresses for the thermal only, mechanical only and combined load cases. Determine also the magnitude and location of the maximum von Mises stress for each load case.
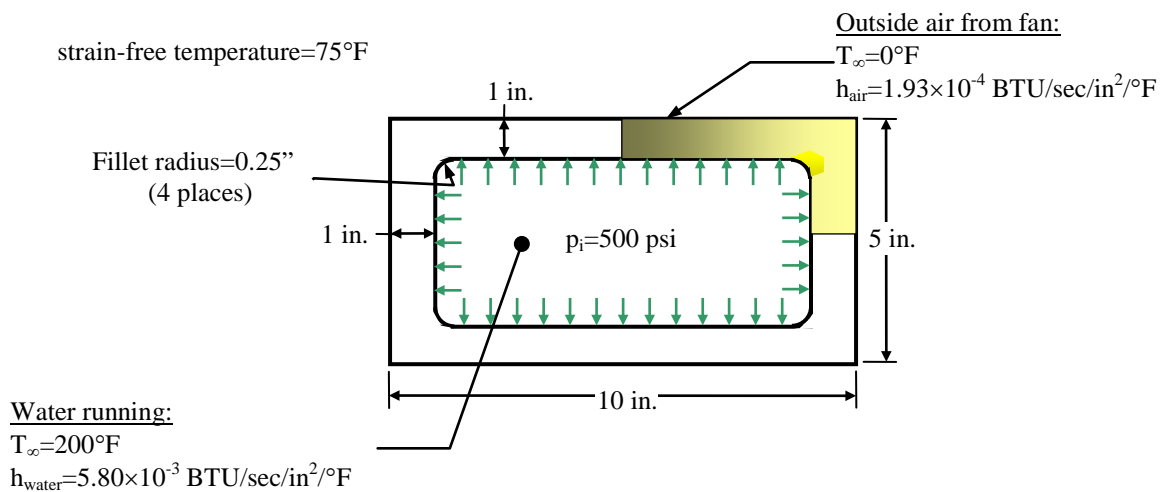
Note that, because of symmetry, only one-quarter of the section needs to be modeled, as shown by the shaded portion of the illustration above. Since the conduit is long in a direction perpendicular to the paper, plane strain conditions will be imposed on the model.

Create the mesh using free meshing. The finite element mesh, loads and BC's will look as follows:

The results for the maximum von Mises stress are:

Load Case #	Analysis Type	$(\sigma_{VM})_{max}$
1	Thermal only	28,515 psi
2	Mechanical only	24,541 psi
3	Combined	31,229 psi

GENERAL COMMANDS IN ANSYS FOR THERMAL STRESS CASE STUDY WITH COMBINED TEMPERATURE AND PRESSURE LOADING

1. Start ANSYS "Mechanical APDL 15.0"

2. Change working directory from default to your own working directory
    Utility Menu > File > Change Directory

3. Change jobname from default (default is "file")
    Utility Menu > File > Change Jobname

    Note: When working with your model, make sure to save the database often
    Utility Menu > File > Save as Jobname.db, or
    Toolbar > Save Analysis (floppy disk icon), or
    Toolbar > SAVE_DB

4. For clarity, you may want to tell ANSYS what type of analysis you want to see menu choices for by using the "Preferences" command. Choose "Structural" and "Thermal" for this problem to show only these disciplines in the GUI.
    Main Menu > Preferences → Structural, Thermal

5. Initially, we will run only the heat transfer analysis, which will provide the temperature distribution in the conduit. Then, later, we will create a new database, by copying the original database, in order to apply the thermal and pressure loading, and combine these loads.

PREPROCESSOR (PREP7 module)

6. Define element type
    Main Menu > Preprocessor > Element Type > Add/Edit/Delete → Add → Thermal Solid → Quad 8 node 77. Use K1="Consistent", and K3="Plane" under "Options".

7. Define real constants
    The plane option does not require real constants

8. Define the thermal conductivity using the (**Main Menu > Preprocessor > Material Props > Material Models → Thermal → Conductivity → Isotropic)** command
    (Material Model Number 1 [steel] → KXX=4.63×10⁻⁴ BTU/sec/in/°F)

9. Create a geometry appropriate to mesh later with free meshing.
   a) Create two rectangles defining the one-quarter geometry of the conduit: **Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners**
   b) Add the two areas using Boolean operations: **Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Areas** (this command will create a single area)
   c) Create lines associated with 0.25-in. radius fillet: **Main Menu > Preprocessor > Modeling > Create > Lines > Line Fillet** (choose intersecting lines and fillet radius)
   d) Create an area from the three lines associated with the fillet: **Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > By Lines**
   e) Add the resulting areas: **Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Areas** (this command will create a single area including the fillet)

10. Mesh the area (create nodes and elements)

    Use the mesh tool to define the mash attributes: **Main Menu > Preprocessor > Meshing > Mesh Tool**
    a) Under "Element Attributes" leave as the default "Global"
    b) In "Size Controls" set "Areas" and set the element edge length SIZE=0.2 in.
    c) In the block under "Mesh", select Mesh: Areas, Shape: Quad, Free radio button
    d) In order to mesh the area, click on the "Mesh" button and select the area
    e) The area is now meshed, but you will notice that the element size is coarse near the fillet. In order to refine near the fillet, choose "Refine at: Lines", click the "Refine" button, and choose the line on the fillet. Use LEVEL=1 (Minimal), which should provide enough refinement (LEVEL=1 cuts the element size approximately in half).
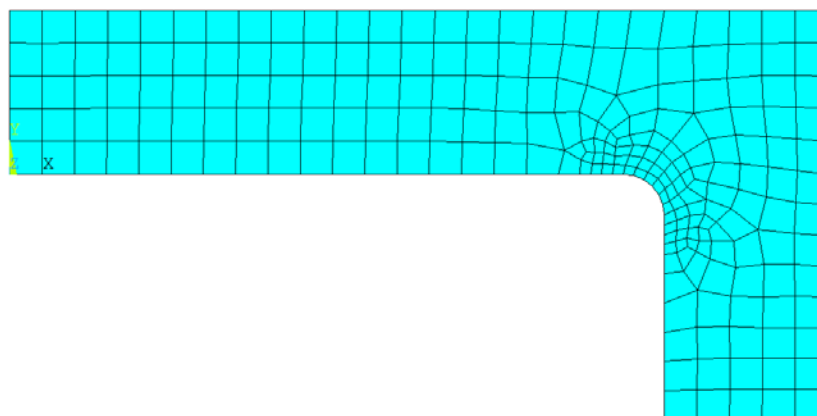
11. Specify the convection conditions: **(Main Menu > Preprocessor > Loads > Define Loads > Apply > Thermal > Convection > On Lines** → *Apply proper convection conditions for water running and outside air*)

    Note that the "Loads" menu can be found under the "Solution" menu as well:
    Main Menu > Solution > Define Loads

12. Specify the symmetry boundary conditions for heat transfer by specifying insulated boundaries (zero heat flux) on the planes of symmetry:
   **Main Menu > Preprocessor > Loads > Define Loads > Apply > Thermal > Heat Flux > On Lines** → *Apply zero heat flux on planes of symmetry*

13. Note that applying the thermal loads at the lines actually means applying the conditions to the *elements* with edges at the line locations. This is done automatically upon initiation of the solution calculations; however, you can also use the following command:
    Main Menu > Preprocessor > Loads > Define Loads > Operate > Transfer to FE > All Solid Lds

14. Use **(Main Menu > Solution > Solve > Current LS)** to run the heat transfer problem. As part of the solution, the program will create a results file in the working directory, ***Jobname.RTH*** (**R**esults **TH**ermal Analysis).

    If the solution completes successfully, you will see the following message
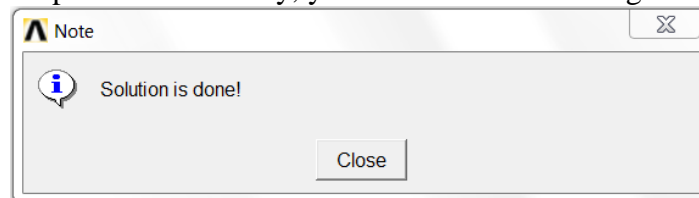
    If you see a different message, it is likely that there was an error and the solver did not complete successfully. See the ANSYS Output Window for details if you get an error.

15. Use (**Main Menu > General Postproc > Plot Results >Contour Plot >Nodal Solu** → **DOF Solution** → **Nodal Temperature**) to view nodal temperature distribution contours. Alternatively, you can use (**Main Menu > General Postproc > Results Viewer**) to view temperature distribution contours.

    Note that the heat transfer analysis does not provide either displacements or stresses but only temperatures at each of the nodes in the model.

APPLICATION OF TEMPERATURE AND PRESSURE LOADING
16. We will create two different load cases with the temperature distribution resulting from the heat transfer analysis and the mechanical (pressure) loading in ANSYS, and later combine them. We will run the thermal and pressure loading analyses as two separate load cases (LC): 1) Thermal load only, and 2) Pressure load only. Then, we will combine them into a third load case.

17. At this point, we will create a new database, define a reference temperature, and apply displacement boundary conditions:

    a) First, make a copy of the database using **(Utility Menu > File > Save As)** and save under a different name from that of the heat transfer analysis. We will use this new database to solve the thermal stress and mechanical load analyses, and combine them.

    b) Exit the current database and open the new one. In the new database, use **(Main Menu > Preprocessor > Loads > Define Loads > Delete > All Load Data > All Loads & Opts)** to delete all heat transfer analysis load conditions.

    c) Use **(Main Menu > Preprocessor > Element Type > Switch Elem Type > Thermal to Struc)** to change the element from thermal to the corresponding structural element. Make sure also to set the proper options for the structural element type selected (K1="Quadrilateral", K3="Plane strain", and K6="Pure displacement" under "Options").

    d) Define real constants: The plane strain option does not require real constants

    e) Define material properties for the structural element. These properties should include Young's modulus (EX), Poisson's ratio (PRXY) and secant coefficient of thermal expansion (ALPX). The thermal conductivity KXX may already be defined from the heat transfer analysis, but it is not necessary for the structural analysis and could be deleted.
    Main Menu > Preprocessor > Material Props > Material Models → Structural → Linear → Elastic → Isotropic
    (Material Model Number 1 [steel] → EX=30e6 psi, PRXY=0.29)
    Main Menu > Preprocessor > Material Props > Material Models → Structural → Thermal Expansion → Secant Coefficient → Isotropic
    (Material Model Number 1 [steel] → ALPX=8.0e-6 in/in/°F)

    f) Use **(Main Menu > Preprocessor > Loads > Define Loads > Settings > Reference Temp → TREF=75°F)** in order to define the reference temperature for zero strain calculations. Temperature differentials for stress and strain calculation will be obtained from this temperature. Thermal strains are given by $\varepsilon = \alpha \, (T - T_{ref})$.

    g) Apply the displacement boundary conditions to the structural model using **(Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement → On Lines →** *Apply symmetry BS's on planes of symmetry***)**.

    h) Transfer all BC's to FE. This step is optional since this is done automatically upon initiation of the solution calculations.
    Main Menu > Preprocessor > Loads > Define Loads > Operate > Transfer to FE > All Solid Lds

18. We will run the thermal and pressure loading analyses as two separate load cases (LC): 1) Thermal stress only, and 2) Mechanical load only. Then, we will combine them into a third load case. First, we will create the applied loads only (18a-b), and we will then run the model simultaneously for these two load cases (19a-b).

    a) Create LC 1, the thermal loading only, by reading the nodal temperatures computed during the thermal analysis step. Use **(Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Temperature > From Therm Analy)**. You can leave LSTEP, SBSTEP and TIME blank and select the results files from the heat transfer analysis, ***Jobname.RTH***. You can check the nodal temperatures read by using **(Utility Menu > List > Loads > Body > On All Nodes)**. Now, create a

temperature load case loading to be solved later using **(Main Menu > Preprocessor > Loads > Load Step Opts > Write LS File → *if you leave LSNUM blank, it defaults to the last load case defined +1, or 1 in this case*)**. The last command creates a load step file named ***Jobname.Sn***, where n=load step number (n=1 in this case).

b) In order to create LC 2, first delete all temperature loads using **(Main Menu > Preprocessor > Loads > Define Loads > Delete > Structural > Temperature > On Nodes > Pick All)**. The displacement boundary conditions should be left unchanged. Now apply the mechanical loading (pressure in this case) using **(Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure → On Lines)**. You can check the pressures applied by using **(Utility Menu > List > Loads > Surface > On All Lines)**. Create a mechanical load case loading to be solved later using **(Main Menu > Preprocessor > Loads > Load Step Opts > Write LS File →** *if you leave LSNUM blank, it defaults to the last load case defined +1, or 2 in this case*)**. The last command creates a load step file named ***Jobname.Sn***, where n=load step number (n=2 in this case). Again, this is a text file with the load case information, *Jobname.Sn*, which you can open and view in your directory.

c) Optional: You can use **Main Menu > Preprocessor > Loads > Define Loads > Operate > Transfer to FE > All Solid Lds** in order to see the pressures applied to the elements, not just lines.

SOLVER (SOLUTION module)

19. We now will run the model simultaneously for these two load cases:
   a) Read and solve the two load cases created by using **(Main Menu > Solution > Solve > From LS Files)** to run the thermal stress problem. Choose load step file numbers 1 and 2 to be read and solved (these load step files are ***Jobname.S1*** and ***Jobname.S2***). Use LSMIN=1, LSMAX=2 and LSINC=1.

   If the solution completes successfully, you will see the following message
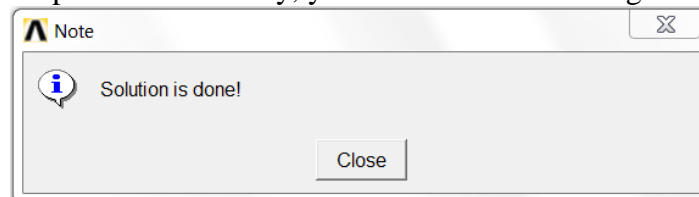
   If you see a different message, it is likely that there was an error and the solver did not complete successfully. See the ANSYS Output Window for details if you get an error.

   b) The solution in a) will create a results file ***Jobname.RST*** (**R**esults **ST**ructural) containing two sets of results (two load steps) corresponding to the two load cases solved (the file is binary and you will not be able to read it with a text editor). Load step 1 will correspond to the thermal stress analysis, while load step 2 to the mechanical stress analysis. You can check the results available in the result file by **(Main Menu > General Postproc > Results Summary)**. Also, you can view displacement and stress results for these two load cases in the results viewer, **(Main Menu > General Postproc > Results Viewer)**. You can switch between load steps using the slider bar in the results viewer. Alternative, you can plot the results using **(Main Menu > General Postproc > Plot Results)**.

POST-PROCESSOR (POST1 module)

20. We will now combine the two load cases solved using superposition to create a third, combined, load case. This third load case will be a linear combination of the first two load cases.

    a) Create two load cases using the **(Main Menu > General Postproc > Load Case > Create Load Case > Results File →** Use a reference number of LCNO=1 for load step LSTEP=1 and LCNO=2 for load step LSTEP=2 to avoid confusion**)**. These two load cases (1 and 2), which will be superimposed later, point to a matching load step in the results file, ***Jobname.RST***. If desired, each load case can be scaled by a factor later when read into the database by using **(Main Menu > General Postproc > Load Case > Calc Options > Scale Factor)**.

    b) Zero out the results portion of the database, **(Main Menu > General Postproc > Load Case > Zero Load Case)**. We will now read the first load case to be superimposed (LC 1) into the database: **(Main Menu > General Postproc > Load Case > Read Load Case → LCNO=1)**. Load case 2 can next be added by using **(Main Menu > General Postproc > Load Case > Add → LCASE1=2)**.

    c) The database now contains one combined load case which is the superposition of cases 1 and 2 (each times a factor if applicable). You can check the status of the load cases created (1 and 2) from **(Main Menu > General Postproc > Load Case > List Load Cases → List All)**.

    d) We will now append the newly created load case (which at this point only exists in the database) to the results file, ***Jobname.RST***, by the command **(Main Menu > General Postproc > Write Results →** call it load step LSTEP=3 with time associated of TIME=3; the latter is arbitrary and could be any number**)**. You can check the load step results that you have in the ***Jobname.RST*** results file by the command **(Main Menu > General Postproc > Results Summary)**. You should see three load steps: 1, 2 and 3. These load steps can be viewed in the results viewer, **(Main Menu > General Postproc > Results Viewer)**. You can switch between load steps using the slider bar in the results viewer. Alternative, you can plot the results using **(Main Menu > General Postproc > Plot Results)**.

21. Before exiting, always save a copy of the command log file from the database log as a backup:
    Utility Menu > File > Write DB Log File → Jobname.lgw