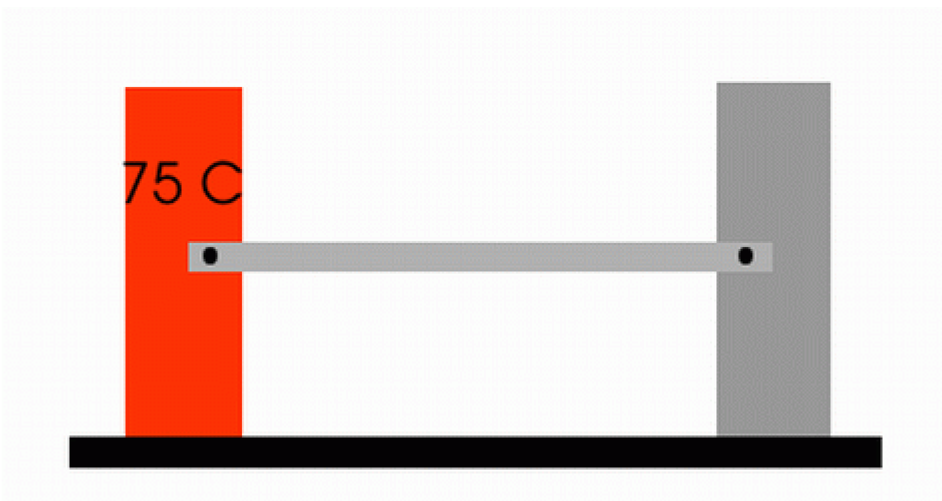
Coupled Structural/Thermal Analysis

Introduction

This tutorial was completed using ANSYS 7.0 The purpose of this tutorial is to outline a simple coupled thermal/structural analysis. A steel link, with no internal stresses, is pinned between two solid structures at a reference temperature of 0 C (273 K). One of the solid structures is heated to a temperature of 75 C (348 K). As heat is transferred from the solid structure into the link, the link will attemp to expand. However, since it is pinned this cannot occur and as such, stress is created in the link. A steady-state solution of the resulting stress will be found to simplify the analysis.

Loads will not be applied to the link, only a temperature change of 75 degrees Celsius. The link is steel with a modulus of elasticity of 200 GPa, a thermal conductivity of 60.5 W/m*K and a thermal expansion coefficient of 12e-6 /K.



Preprocessing: Defining the Problem

According to Chapter 2 of the ANSYS Coupled-Field Guide, "A sequentially coupled physics analysis is the combination of analyses from different engineering disciplines which interact to solve a global engineering problem. For convenience, ...the solutions and procedures associated with a particular engineering discipline [will be referred to as] a physics analysis. When the input of one physics analysis depends on the results from another analysis, the analyses are coupled."

Thus, each different physics environment must be constructed seperately so they can be used to determine the coupled physics solution. However, it is important to note that a single set of nodes will exist for the entire model. By creating the geometry in the first physical environment, and using it with any following coupled environments, the geometry is kept constant. For our case, we will create the geometry in the Thermal Environment, where the thermal effects will be applied.

Although the geometry must remain constant, the element types can change. For instance, thermal elements are required for a thermal analysis while structural elements are required to deterime the stress in the link. It is important to note, however that only certain combinations of elements can be used for a coupled physics analysis. For a listing, see Chapter 2 of the ANSYS Coupled-Field Guide located in the help file.

The process requires the user to create all the necessary environments, which are basically the preprocessing portions for each environment, and write them to memory. Then in the solution phase they can be combined to solve the coupled analysis.

Thermal Environment - Create Geometry and Define Thermal Properties

1. **Give example a Title**
   Utility Menu > File > Change Title ...
   `/title, Thermal Stress Example`

2. **Open preprocessor menu**
   ANSYS Main Menu > Preprocessor
   `/PREP7`

3. **Define Keypoints**
   Preprocessor > Modeling > Create > Keypoints > In Active CS...
   `K,#,x,y,z`

   We are going to define 2 keypoints for this link as given in the following table:

Keypoint	Coordinates (x,y,z)
1	(0,0)
2	(1,0)

4. **Create Lines**
   Preprocessor > Modeling > Create > Lines > Lines > In Active Coord
   `L,1,2`

   Create a line joining Keypoints 1 and 2, representing a link 1 meter long.

5. **Define the Type of Element**
   Preprocessor > Element Type > Add/Edit/Delete...

   For this problem we will use the LINK33 (Thermal Mass Link 3D conduction) element. This element is a uniaxial element with the ability to conduct heat between its nodes.

6. **Define Real Constants**
   Preprocessor > Real Constants... > Add...

   In the 'Real Constants for LINK33' window, enter the following geometric properties:
   i. Cross-sectional area AREA: 4e-4

   This defines a beam with a cross-sectional area of 2 cm X 2 cm.

7. **Define Element Material Properties**
   Preprocessor > Material Props > Material Models > Thermal > Conductivity > Isotropic

   In the window that appears, enter the following geometric properties for steel:
   i. KXX: 60.5

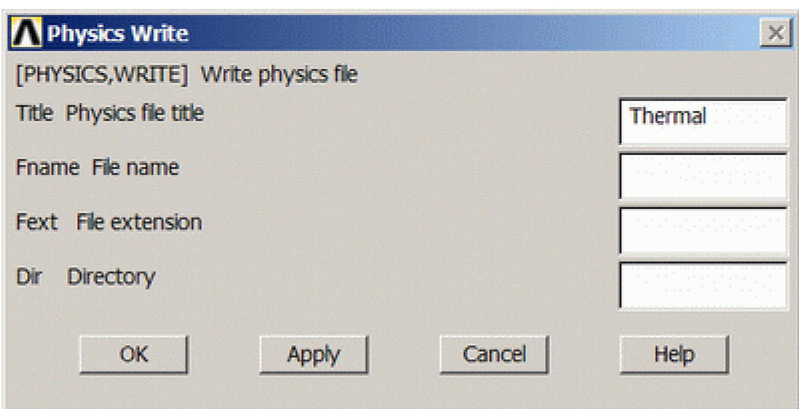
8. **Define Mesh Size**
   Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines...

   For this example we will use an element edge length of 0.1 meters.

9. **Mesh the frame**
   Preprocessor > Meshing > Mesh > Lines > click 'Pick All'

10. **Write Environment**
    The thermal environment (the geometry and thermal properties) is now fully described and can be written to memory to be used at a later time.
    Preprocessor > Physics > Environment > Write

    In the window that appears, enter the TITLE **Thermal** and click OK.



11. **Clear Environment**
    Preprocessor > Physics > Environment > Clear > OK

    Doing this clears all the information prescribed for the geometry, such as the element type, material properties, etc. It does not clear the geometry however, so it can be used in the next stage, which is defining the structural environment.

Structural Environment - Define Physical Properties

Since the geometry of the problem has already been defined in the previous steps, all that is required is to detail the structural variables.

1. **Switch Element Type**
   Preprocessor > Element Type > Switch Elem Type

   Choose **Thermal to Struc** from the scoll down list.

This will switch to the complimentary structural element automatically. In this case it is LINK 8. For more information on this element, see the help file. A warning saying you should modify the new element as necessary will pop up. In this case, only the material properties need to be modified as the geometry is staying the same.

2. **Define Element Material Properties**
   Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

   In the window that appears, enter the following geometric properties for steel:
   i. Young's Modulus EX: 200e9
   ii. Poisson's Ratio PRXY: 0.3

   Preprocessor > Material Props > Material Models > Structural > Thermal Expansion Coef > Isotropic
   i. ALPX: 12e-6

3. **Write Environment**
   The structural environment is now fully described.
   Preprocessor > Physics > Environment > Write

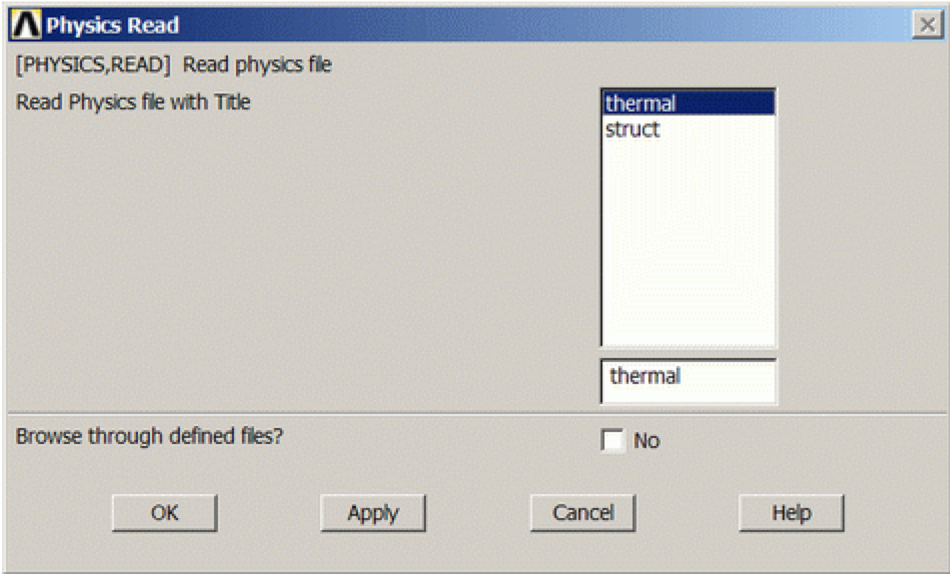
   In the window that appears, enter the TITLE **Struct**

Solution Phase: Assigning Loads and Solving

1. **Define Analysis Type**
   Solution > Analysis Type > New Analysis > Static
   `ANTYPE,0`

2. **Read in the Thermal Environment**
   Solution > Physics > Environment > Read

   Choose **thermal** and click OK.



If the Physics option is not available under Solution, click **Unabridged Menu** at the bottom of the Solution menu. This should make it visible.

3. **Apply Constraints**
   Solution > Define Loads > Apply > Thermal > Temperature > On Keypoints

   Set the temperature of Keypoint 1, the left-most point, to 348 Kelvin.

4. **Solve the System**
   Solution > Solve > Current LS
   `SOLVE`

5. **Close the Solution Menu**
   Main Menu > Finish

   It is very important to click **Finish** as it closes that environment and allows a new one to be opened without contamination. If this is not done, you will get error messages.

   The thermal solution has now been obtained. If you plot the steady-state temperature on the link, you will see it is a uniform 348 K, as expected. This information is saved in a file labelled `Jobname.rth`, were .rth is the thermal results file. Since the jobname wasn't changed at the beginning of the analysis, this data can be found as **file.rth**. We will use these results in determing the structural effects.

6. **Read in the Structural Environment**
   Solution > Physics > Environment > Read

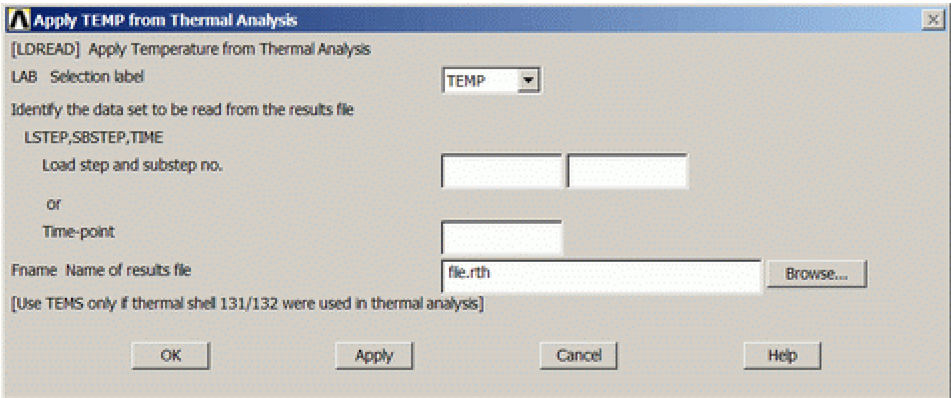
   Choose **struct** and click OK.

7. **Apply Constraints**
   Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

   Fix Keypoint 1 for all DOF's and Keypoint 2 in the UX direction.

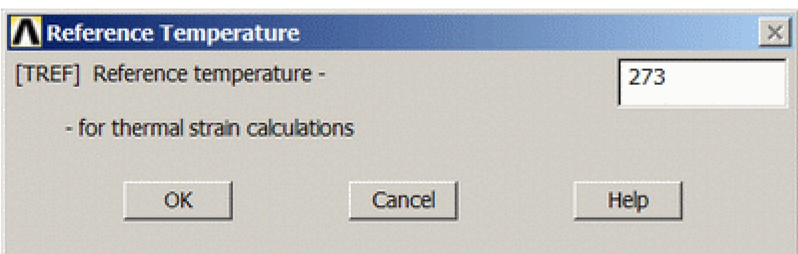
8. **Include Thermal Effects**
   Solution > Define Loads > Apply > Structural > Temperature > From Therm Analy

   As shown below, enter the file name `File.rth`. This couples the results from the solution of the thermal environment to the information prescribed in the structural environment and uses it during the analysis.



9. **Define Reference Temperature**
   Preprocessor > Loads > Define Loads > Settings > Reference Temp

   For this example set the reference temperature to 273 degrees Kelvin.



10. **Solve the System**
    Solution > Solve > Current LS
    `SOLVE`

Postprocessing: Viewing the Results

1. **Hand Calculations**
   Hand calculations were performed to verify the solution found using ANSYS:

Expansion due to thermal stress in a link can be calculated using:

$$\delta = \alpha \Delta T L$$

Expansion due to structural forces can be determined using:

$$\delta = \frac{PL}{EA}$$

Soving for the structural forces due to the thermal expansion,

$$P = \alpha \Delta T E A$$

Or

$$\sigma = \frac{F}{A} = \alpha \Delta T E$$

Therefore, in this example

$$\sigma = (0.000012/\mathrm{K})(348\ \mathrm{K} - 273\ \mathrm{K})(200e3\ \mathrm{MPa}) = 180\ \mathrm{MPa}$$

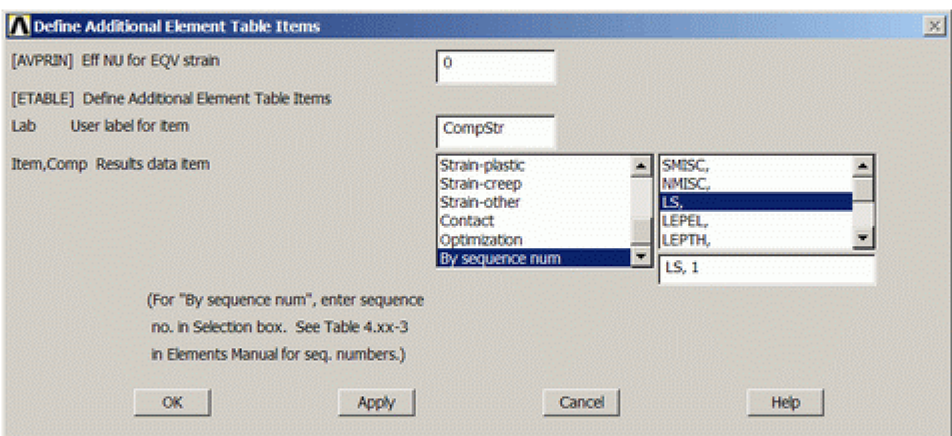
As shown, the stress in the link should be a uniform 180 MPa in compression.

2. **Get Stress Data**

   Since the element is only a line, the stress can't be listed in the normal way. Instead, an element table must be created first.

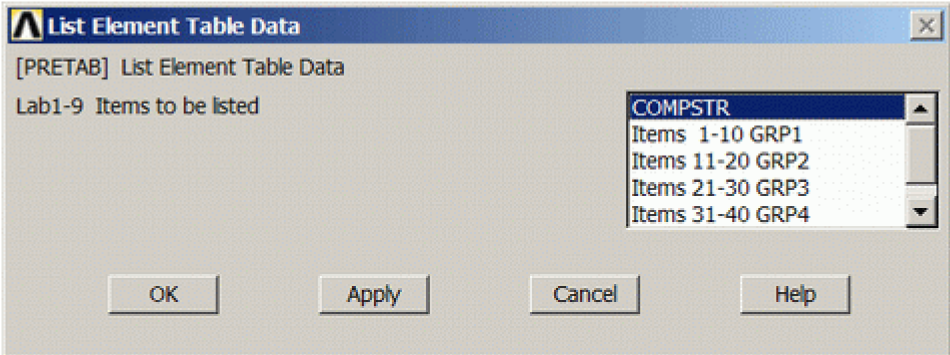
   General Postproc > Element Table > Define Table > Add

   Fill in the window as shown below. [CompStr > By Sequence Num > LS > LS,1
   `ETABLE,CompStress,LS,1`

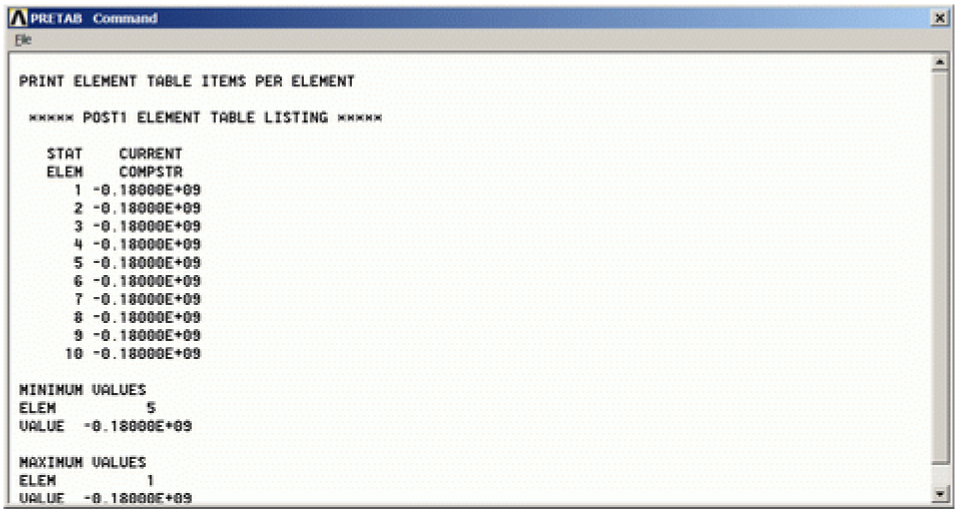


3. **List the Stress Data**

   General Postproc > Element Table > List Elem Table > COMPSTR > OK
   `PRETAB,CompStr`



The following list should appear. Note the stress in each element: -0.180e9 Pa, or 180 MPa in compression as expected.



Command File Mode of Solution

The above example was solved using a mixture of the Graphical User Interface (or GUI) and the command language interface of ANSYS. This problem has also been solved using the ANSYS command language interface that you may want to browse. Open the .HTML version, copy and paste the code into Notepad or a similar text editor and save it to your computer. Now go to **'File > Read input from...'** and select the file. A .PDF version is also available for printing.