Radiation Between Surfaces - Physics Setup

Author: Chia-Hsun Hsieh, Cornell University

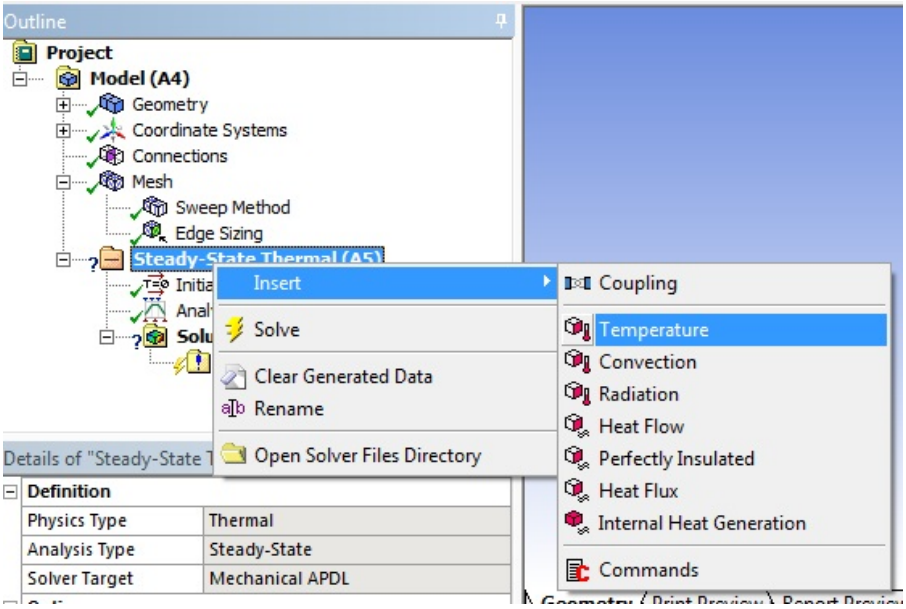
Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
Exercises
Comments

Physics Setup

Set-up Initial Conditions

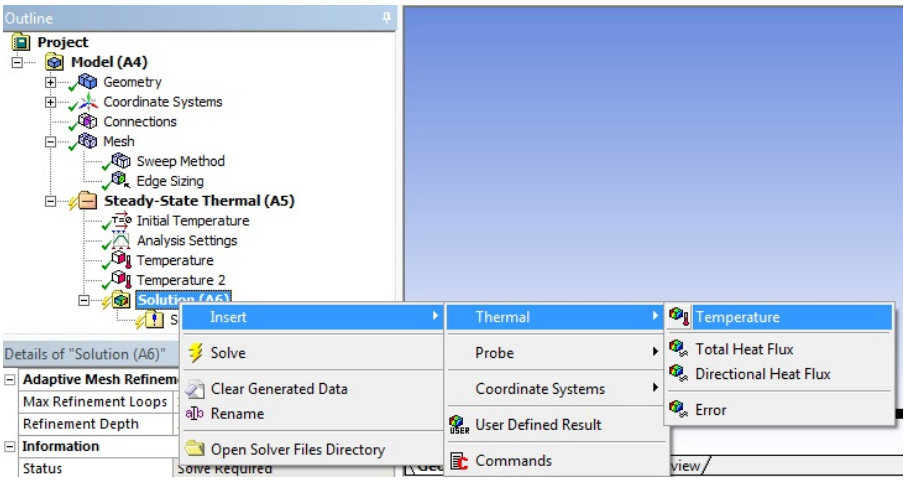
Steady-State Thermal

We will need to run the steady state model and use the result as the initial condition for the transient analysis.
Right click on ***Steady-State Thermal (A5)*** > ***Insert*** > ***Temperature***.

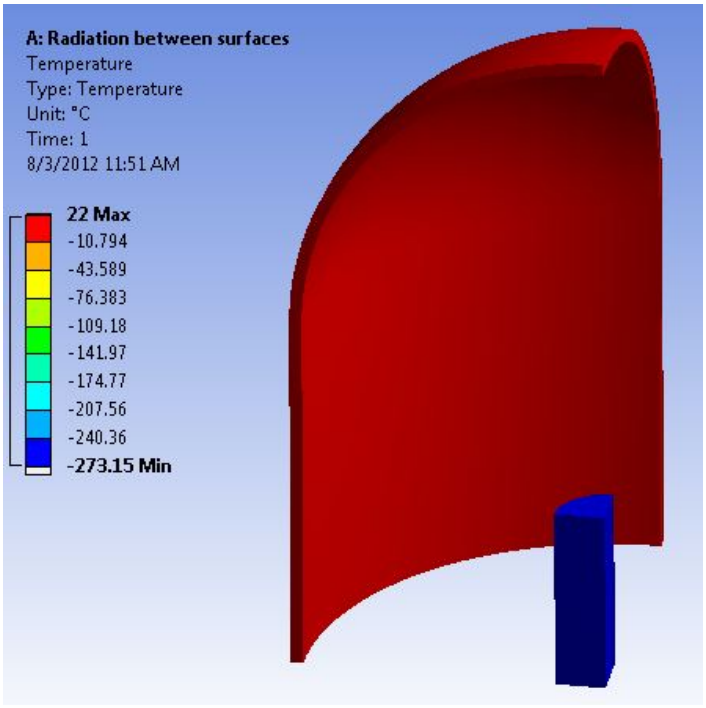


Select the entire ***Shell*** body and set the temperature to ***22 degrees Celsius***. Create another temperature boundary condition but select the ***Specimen*** instead. Set the temperature of the ***Specimen*** to ***-273.15 degrees Celsius***.

Right click on ***Solution (A6)*** > ***Insert*** > ***Thermal*** > ***Temperature***. The default geometry is set to ***All Bodies***. Keep it and repeat the step but select only the ***Specimen***.



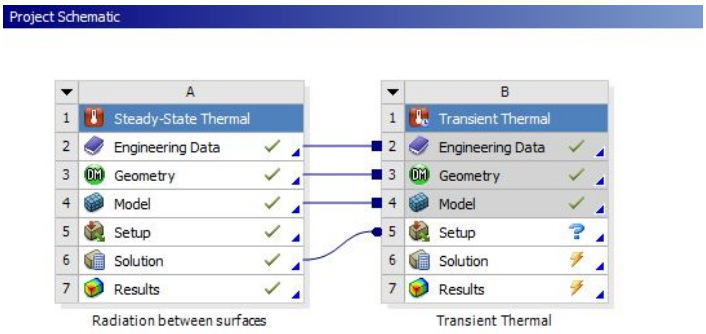
The solution titled ***Temperature*** will display the temperature distribution of the shell and the specimen and ***Temperature 2*** will display only the specimen. Notice there isn't any temperature variation because we have done nothing except set the temperature of the two bodies. No heat can be exchanged between the two bodies without specifying additional boundary conditions (convection, radiation, etc).



We are now ready to move on to set up the transient analysis.

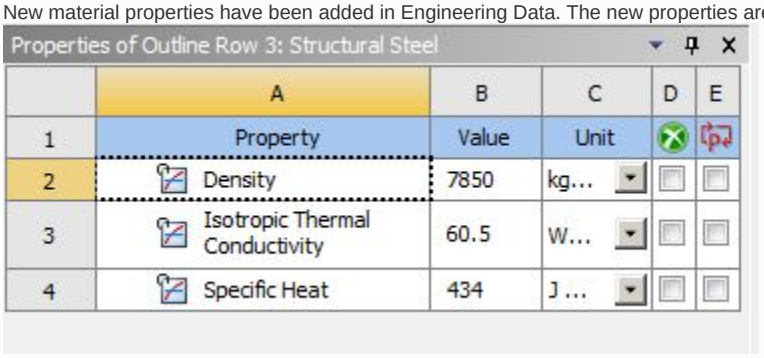
Set-up Transient Thermal Analysis

Return to the ***Project Schematic*** in ANSYS Workbench. Right click on ***Solution*** > ***Transfer Data to New*** > ***Transient Thermal***. This will export the model, the mesh, and the steady state solution to ***Transient Thermal*** analysis and the new analysis is ready to be set-up.



Additional Material Properties

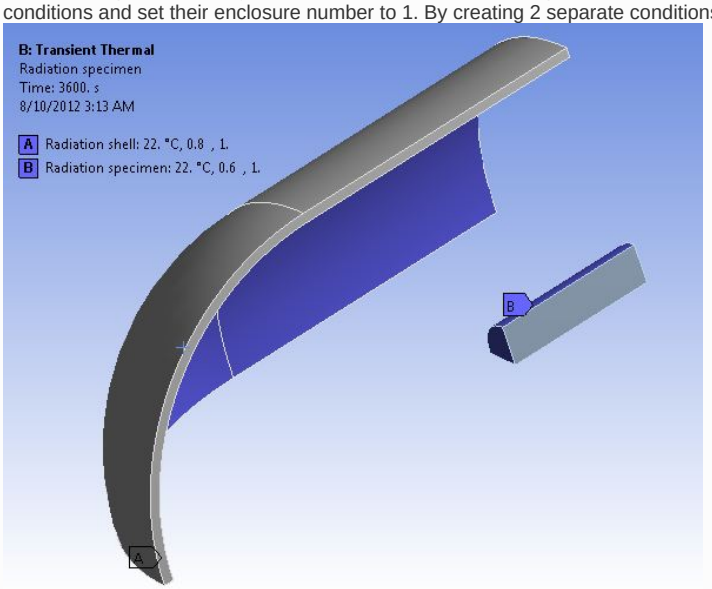
New material properties have been added in Engineering Data. The new properties are essential to perform transient thermal analysis.



Please go through the following video to specify the symmetry regions in the model. This will allow the FEA code to compute the ***View Factor*** between the surface of the shell and the specimen. The View Factor is an important parameter in radiation analysis.

Surface to Surface Radiation

Surface to surface radiation is applied like a boundary condition. Radiating surfaces are related to one another by their enclosure number. We want to set up the boundary condition to make the shell and specimen surface to "see" one another. This can be done by creating 2 radiation conditions and set their enclosure number to 1. By creating 2 separate conditions, each surface can have different emissivity value.



Details of "Radiation shell"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Radiation
Correlation	Surface to Surface
Emissivity	0.8 (step applied)
Ambient Temperature	22. °C (step applied)
Enclosure	1.
Suppressed	No

Proceed to the following video to set up the thermal boundary conditions.

Once the Convection and Radiation boundary conditions have been set up, you may move on to the next step to set up the solution.

Go to Step 5: Numerical Solution

Go to all ANSYS Learning Modules