Radiation Between Surfaces - Physics Setup

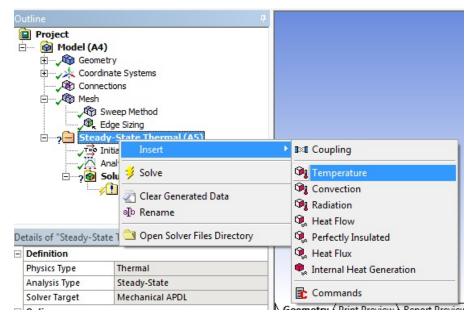
Author: Chia-Hsun Hsieh, Cornell University Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Numerical Solution 6. Numerical Results 7. Verification & Validation Exercises Comments

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*.

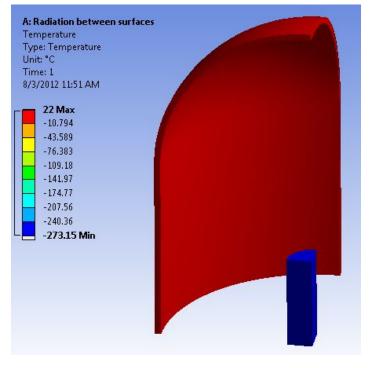


Select the entire *Shell* body and set the temperature to *22 degrees Celsius*. Create another temperature boundary condition but select the *Specimen* instead. Set the temperature of the *Specimen* to *-273.15 degrees Celsius*.

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.

Outline Project Model (A4) Model (A4) Mesh Coordinate S Coordinate	Method izing ate Thermal (A5) emperature is Settings rature rature rature 2		
		Thermal	• 🗣 Temperature
Details of "Solution (A6)"	誟 Solve	Probe	Total Heat Flux
Adaptive Mesh Refinem	Clear Generated Data	Coordinate Systems	Directional Heat Flux
Max Refinement Loops	allo Rename		🍬 Error
Refinement Depth	-	See User Defined Result	
Information	Open Solver Files Directory	Commands	time (
Status	Solve Required	Commands	view/

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.

•		A			-	В		
1		Steady-State Thermal			1	Transient Thermal		
2	0	Engineering Data	~	4	2 🥥	Engineering Data	~	4
3	00	Geometry	1	-	3 🕅	Geometry	~	
4	0	Model	~	4	4 🥩	Model	~	4
5		Setup	~	4	5	Setup	?	-
6	6	Solution	~	-	6	Solution	7	
7	1	Results	1		7 😥	Results	7	

Additional Material Properties

Pro

()

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

	А	В	С	D	E
1	Property	Value	Unit	8	tp.
2	🔁 Density	7850	kg 💌		
3	Conductivity	60.5	w 💌		
4	🔁 Specific Heat	434	J 💌		

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.

con	nt to set up the boundary conditio	on to make the shell and spec	Radiating surfaces are related to one another by their enclosure number. imen surface to "see" one another. This can be done by creating 2 radiat arate conditions, each surface can have different emissivity value.
Ra Tir	Transient Thermal diation specimen me: 3600. s 10/2012 3:13 AM	\square	
	Radiation shell: 22. °C, 0.8 , 1. Radiation specimen: 22. °C, 0.6 , 1.		
De	tails of "Radiation shell"		7
-	etails of "Radiation shell" Scope		7
		Geometry Selection	P
	Scope	Geometry Selection 2 Faces	P
	Scope Scoping Method		P
	Scope Scoping Method Geometry		P
	Scope Scoping Method Geometry Definition	2 Faces	
	Scope Scoping Method Geometry Definition Type	2 Faces Radiation	
	Scope Scoping Method Geometry Definition Type Correlation Emissivity	2 Faces Radiation Surface to Surface	
	Scope Scoping Method Geometry Definition Type Correlation Emissivity	2 Faces Radiation Surface to Surface 0.8 (step applied)	

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

 ~

B B (1)

Go to all ANSYS Learning Modules