Thermal Stresses in a Bar - Mesh

Author: Ben Mullen, 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

Mesh

Open the Solver



Set the mesh size

When the solver opens, click on methods in the Outline window to open the Mesh menu in the menu bar. In the Details window, expand Sizing, and change the Element Size from Default to 0.5 m (Make sure the units are set to meters by selecting Units > Metric (m, kg, N, s, V, A) in the menu bar)

Generate the Mesh

When the sizing has been specified, press Mesh > Generate Mesh to create the mesh.

😮 Multiple Systems - Mechanic	al (AN	YS Academic Teaching Advanced)	
File Edit View Units Tool	s Hel	🥥 🦻 Solve = 🎲 📷 🖄 🖄 🐨 🕻	Worksheet.
マケマ い 図 図	1	8. 5 ↔ @ @ @ @ Q 0	₩ /2 음 □•
		Mesh Control + Metric Graph Dptions	
Outline	3		-
🔅 - 🎯 Model (A4, B4)	4	Mech	
B- B Geometry	100	20/9/2011 #28 PM	
E Coordinate System	6 EI		
Steady-State T			
- Thital Tenpe			
Analysis Sett			
4 1	8		
Details of "Mesh"	3		
- Defaults	4		
Physics Pref., Mechanical	-11		
Relevance 0	-11		
 Sizing 			
Use Advanc., Off	-11		
Relevance Coarse	1		
Element 0.50 m			
Initial Size Active Assembly			
Smoothing Medium			
Transition Fast			
Span Angle Coarse			w
Minimum E., 5.0 m			t
Inflation	+		×
Section Planes	Ф×		
🟥 🗙 🧆			
		Geometry (Print Preview), Report Preview/	
Press F1 for Help		No Messag No Selection	Metric (m, kg, N, s, V, A) Degr

Go to Step 4: Physics Setup

Go to all ANSYS Learning Modules