

## *An Introduction to COMSOL Multiphysics*

### *Introduction*

A solid undergoes thermal expansion due to the application of heat along with deformation due to applied loads.

### *Model Definition*

Consider a thin cylinder of length 1 m and inner and outer radii 0.2 m & 0.21 m respectively. The cylinder is kept fixed at one end and at the other end a tensile load of 200 kPa is applied at time (t) =3 sec. An edge load of 1000 N/m is applied to the outer rim at the free end at t=6 sec. The fixed end of the cylinder is at 273.15 K (the ambient temperature) and the free end at 274.15 K (all other sides are insulated). The cylinder expands due to the heat flow. The cylinder is made of aluminum.

### **MODEL NAVIGATOR**

1. On the **New** page select **3D** from the **Space dimension** list.
2. In the list of application modes open the **Structural Mechanics Module>Thermal-Structural Interaction** folder and select **Solid, Stress-Strain with Thermal Expansion**.
3. Click **OK**.

### **GEOMETRY MODELING**

Define the model geometry:

1. In the **Draw** menu select **Work-Plane Settings**.
2. Select **x-y z: 0** and click **OK**.
3. Shift-click the **Ellipse/Circle (Centered)** toolbar button.
4. In the **Circle** dialog box type 0.2 in the **Radius** edit field and click **OK** to create circle C1 with default center as (0,0).
5. Create another concentric circle with radius 0.21.
6. Select both geometry objects by pressing Ctrl+A.
7. Click the **Difference** button on the Draw toolbar.
8. In the **Draw** menu select **Extrude**.
9. Under extrusion parameter give **Distance** as 1.
10. Click **OK**.

## PHYSICS SETTINGS

### *Boundary Conditions—General Heat Transfer*

1. Select the General Heat Transfer application mode in the **Multiphysics** menu.
2. Open the **Boundary Settings** dialog box from the **Physics** menu.
3. Select boundary 3 from **Boundary selection** list
4. Select **Temperature** from the **Boundary conditions** list.
5. Type 273.15 in the **T0** (temperature) edit field.
6. Similarly put boundary 4 at 274.15 K.
7. The remaining boundaries are thermally insulated, which is the default boundary condition.
8. Click **OK**.

### *Subdomain Settings—General Heat Transfer*

1. From the **Physics** menu choose **Subdomain Settings**.
2. Select subdomain 1 from the **Subdomain selection** list.
3. Press the **Load** tab and select **Materials>Basic Material Properties>Aluminum**.
4. Click **OK**.

### *Initial Conditions—General Heat Transfer*

1. Click the **Init** tab.
2. Check if the default value in the edit field for **T (t0)** is 273.15.
3. Click **OK**.

### *Application Mode Selection*

In the **Multiphysics** menu switch to Solid, Stress-Strain application mode by selecting it.

### *Boundary Conditions—Solid, Stress-Strain*

1. Open the **Boundary Settings** dialog box from the **Physics** menu.
2. Select boundary 3 from the **Boundary selection** list.
3. Select **Fixed** from the **Constraint Condition** list.
4. Select boundary 4.
5. Click the **Load** tab and enter the expression  $\text{if}(t \geq 3, 2e5, 0)$  in the **Fz** edit field.
6. Click **OK**.

### *Cylindrical Coordinate System*

1. In the **Options** menu select **Coordinate Systems**.
2. Click the **New** tab and click **OK** in the **New Coordinate System** dialog box.
3. Select **Cylindrical coordinate system** in the **Workplane** tab.
4. Click **OK**.

### *Edge Settings—Solid, Stress-Strain*

1. Open the **Edge Settings** dialog box from the **Physics** menu.
2. Select edges 4, 5, 13 and 22 from the **Edge selection** list.
3. Click the **Load** tab and select **Coordinate System 1** from the **Coordinate System** list.
4. Type the expression  $\text{if}(t \geq 6, 1e3, 0)$  in the **Fyl** edit field.
5. Click **OK**.

### *Subdomain Settings—Solid, Stress-Strain*

1. From the **Physics** menu choose **Subdomain Settings**.
2. Select subdomain 1 from the **Subdomain selection** list.
3. Select **Aluminum** from the **Library material** list.
4. Click the **Load** tab and type 273.15 in the **Tempref** edit field.
5. Click **OK**.

## **MESH GENERATION**

1. Click the **Decrease Mesh Size** button in the mesh toolbar.
2. Click the **Mesh All (Swept)** button in the mesh toolbar.

## **COMPUTING THE SOLUTION**

1. Open the **Solver Parameters** dialog box from the **Solve** menu.
2. Select **Time dependent** in the **Solver** list.
3. Type 0:0.1:10 in the **Times** edit field.
4. Select **Direct (PARDISO)** from the **Linear System Solver** list.
5. Click on the **Time Stepping** tab.
6. Select **Intermediate** from the **Time steps taken by solver** list.
7. Click **OK**.
8. Click **Solve** button on the Main toolbar to run the analysis.

## **POSTPROCESSING & VISUALISATION**

1. Open the **Plot Parameters** dialog box from the **Postprocessing** menu.
2. Click the **Subdomain** tab.
3. Select the **Subdomain plot** check box.
4. Select **von Mises Stress** in the **Predefined quantities** list. Similarly plot Strain energy density, Total Displacement & Temperature.
5. You can also plot **Arrow plots** of total heat flux and temperature gradient to visualize the heat flux.
6. Calculate the reaction at the fixed end by integrating the traction forces using **Boundary integration** from the **Postprocessing** menu.
7. Use **Cross-section Plot Parameters** to obtain plots of stress varying with time at a point.

---

*Questions, comments? Contact: [beatnic@aset.psu.edu](mailto:beatnic@aset.psu.edu)*