AIM Thermal Stresses in a Bar - Physics Set-Up

Author(s): Sebastian Vecchi, ANSYS Inc.

Physics Set-Up

Add Structural Conditions

Once having entered the **Physics** task of the workflow, the wall support needs to be defined. This is done by selecting **Add** next to **Structural Conditions**. In the **Add** drop down menu, there is an option for **Support**. Click this to open the **Support** selection menu. Select the face which is going to be attached to the wall, then press the blue '+' button next to Location. Next, open the drop down menu below **Type** and choose **User specified**. Next, under **Degrees of Freedom**, set **Translation X** and **Translation Y** to be **Free** while **Translation Z** is set to **Fixed**. This allows the bar to expand in the X and Y directions while constraining Z.

The other end of the bar needs a displacement constraint in order to keep it from "clipping" into the other wall. Return to the **Physics** task, select **Add** to the right of **Structural Conditions**, and choose **Displacement**. Select the appropriate face, press "+" next to **Location**, and input 0.002 m in the **Translation Z** box. Then, type **Free** into the **Translation X** and **Translation Y** boxes. Using **Free** in these places will prevent the free end from being over constrained and calculate a more accurate stress distribution.

Next to **Structural Conditions**, press **Add** > **Support**, then select one of the cut sides as the **Location** and set the **Type** to **User specified**. Edit the **Translation** drop down menus until there is only one arrow going into our model. This creates a symmetrical constraint support for the shaft which allows it to deform while also not moving its location. Repeat this step for the other cut face.

Add Solid Thermal Conditions

Next, the temperature change needs to be added by selecting entire bar body, adding a **Solid Thermal Condition** > **Temperature**, and setting it to 122 degrees Celsius. Be sure to change to the **Body selection** mode using the toolbar at the top center of the model window. We know that it needs to be set to 122 degrees Celsius because there needs to be a 100 degree increase. The original temperature can be found by going back to the **Physics** section via the workflow, and then selecting **Material Assignments**. In this menu, the **Zero-thermal-strain reference temperature** can be found and used as the starting temperature for the material. The material was automatically assigned to be structural steel, but if a different material was used, the zero-thermal-strain reference temperature would be different and so would the thermal condition for the free end of the bar.

Go to Step 5: Results

Go to all ANSYS AIM Learning Modules