

AIM Thermal Stresses in a Bar - Pre-Analysis

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results
6. Verification & Validation

Pre-Analysis & Start Up

Pre-Analysis

First, we need to know if the expansion of the bar is greater than the free space between the bar and the wall, $x = 0.002$ meters. Below are the equation and the calculated result used for the expansion of the bar.

$$\delta_T = \alpha L \Delta T = (1.2E - 5^{\circ}C^{-1})(5m)(100^{\circ}C) = 0.006m > 0.002m$$

Now that we know that the deformation due to the change in temperature will be greater than the space between the bar and the wall, we know that there will be a stress on the bar. If the wall was not there, the bar would fully deform to the calculated value above. Since the wall does stop the deformation process, the following equation can be used.

$$\delta_T = x + \delta_{\sigma}$$

Where T is the strain contribution from the change in temperature and is the strain contribution from the force imparted by the wall on the bar. This equation can be combined with the first one to make the following.

$$\alpha L \Delta T = x + \frac{PL}{EA}$$

Substituting $\sigma_{xx} = \frac{P}{A}$, we get the solvable equation

$$\alpha L \Delta T = x + \frac{\sigma_{xx} L}{E}$$

Solving for σ_{xx} , we find that

$$\sigma_{xx} = \alpha E \Delta T - \frac{x E}{L}$$

After substituting, we find

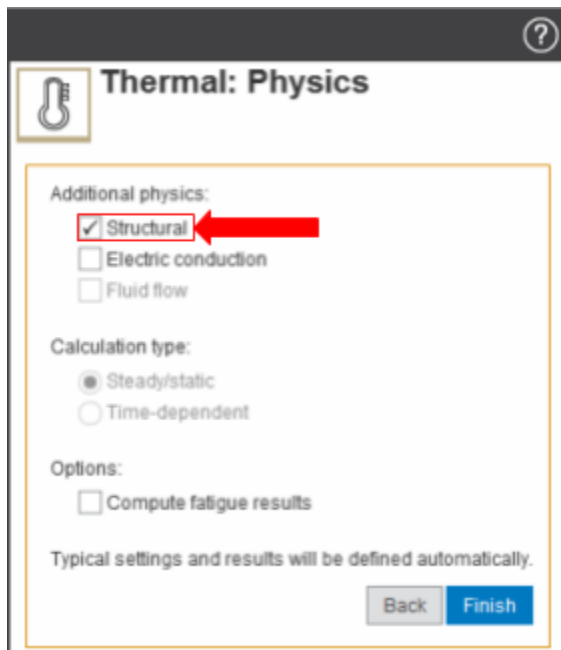
$$\sigma_{xx} = (1.2E - 5^{\circ}C^{-1})(2E11 Pa)(100^{\circ}C) - \frac{(0.002m)(2E11 Pa)}{(5m)} = 160 MPa$$

A few words on the formatting on the following instructions:

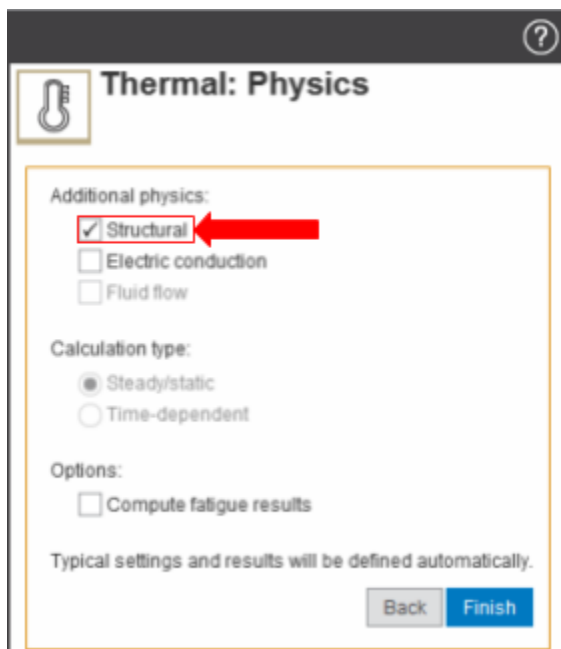
1. Notes that require you to perform an action are colored in blue
2. General information is colored in black, but does not require any action
3. Words that are **bolded** are labels for items found in ANSYS AIM
4. Most important notes are colored in red


Start-Up


Now that we have the pre-calculations, we are ready begin simulating in ANSYS AIM. Open ANSYS AIM by going to **Start > All Apps > ANSYS 18.1 > ANSYS AIM 18.1**. Once you are at the starting page of AIM select the **Thermal** template.



You will be prompted by the **Thermal** template to either **Define new geometry**, **Import geometry file**, or **Connect to active CAD session**. Select **Define new geometry** and press **Next**. For this problem, we will be using the **Steady/static** calculation type and adding **Structural** as an additional physic. Select them in the menu and press **Finish**. The Model Editor will launch automatically.








Thermal: Physics

Additional physics:

☒ Structural 

☐ Electric conduction

☐ Fluid flow

Calculation type:

☒ Steady/static

☐ Time-dependent

Options:

☐ Compute fatigue results

Typical settings and results will be defined automatically.

[Go to Step 2: Geometry](#)

[Go to all ANSYS AIM Learning Modules](#)