

AIM Heat Conduction in a Bar - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

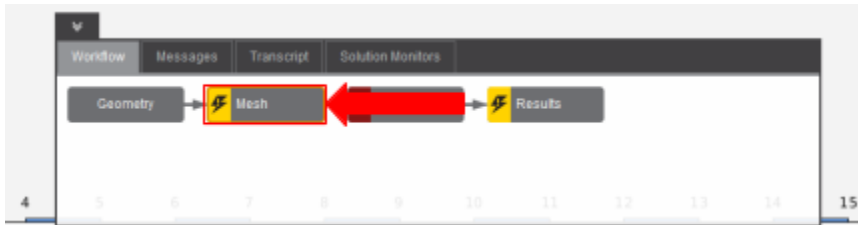
[3. Mesh](#)

[4. Physics Setup](#)

[5. Results](#)

Mesh

Once you have exited the modeling window, [initiate the meshing process by clicking on **Mesh** in the workflow.](#)



Set Mesh Size

Using the **Mesh resolution** slider, we can edit how precise our mesh is for our calculations. [Move the slider all the way to **High**](#) so that we can get the best possible data. [Under **Global Sizing**, select the **Proximity** option for the **Size function method**.](#)

Generate Mesh

[Click **Generate Mesh**](#) under **Output** or at the top of the screen by the status window for **Mesh**. AIM should detect you are ready to generate the mesh and highlight the buttons in blue.

[Go to Step 4: Physics Setup](#)

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