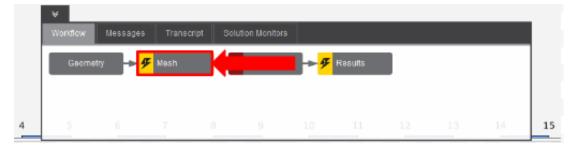
ANSYS Taylor-Couette Flow between Rotating Cylinders -Mesh

Author(s): Sebastian Vecchi, ANSYS Inc

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

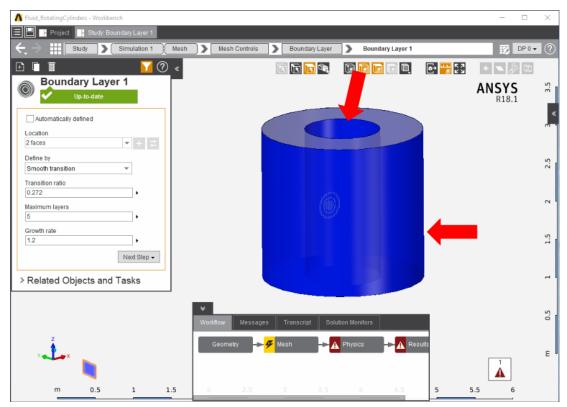
<u>Mesh</u>

Once you have exited the modeling window, initiate the meshing process by clicking on Mesh in the Workflow.



Set Mesh Boundary Layer

AIM will prompt you to fix the boundary layer before generating the mesh. Click on **Boundary Layer** under **Mesh Controls**. Select the inside and outside faces of the fluid flow volume.



Generate Mesh

Return to the Mesh panel, then click Generate Mesh under Output or at the top of the screen by the status window for Mesh. AIM will detect that 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