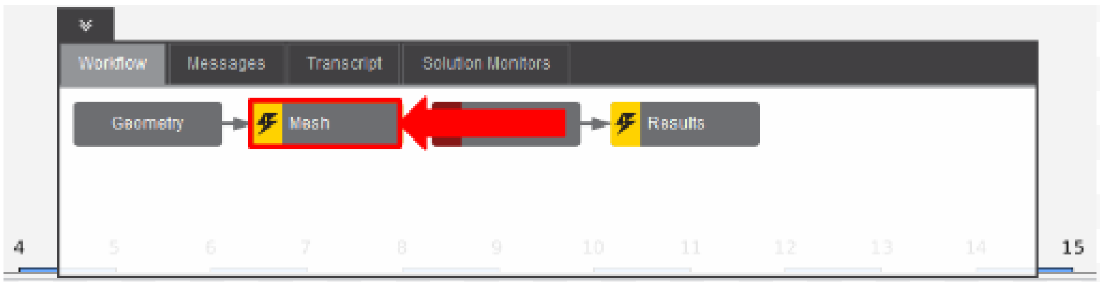
ANSYS Taylor-Couette Flow between Rotating Cylinders - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc

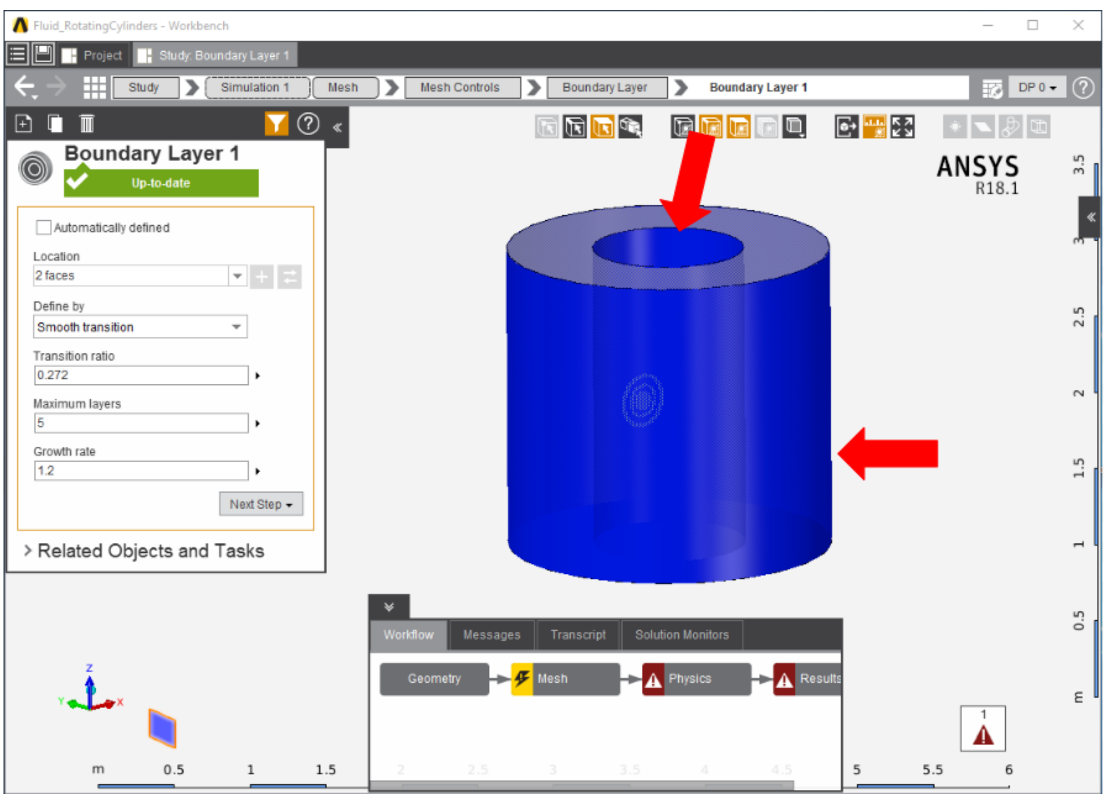
Mesh

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