

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

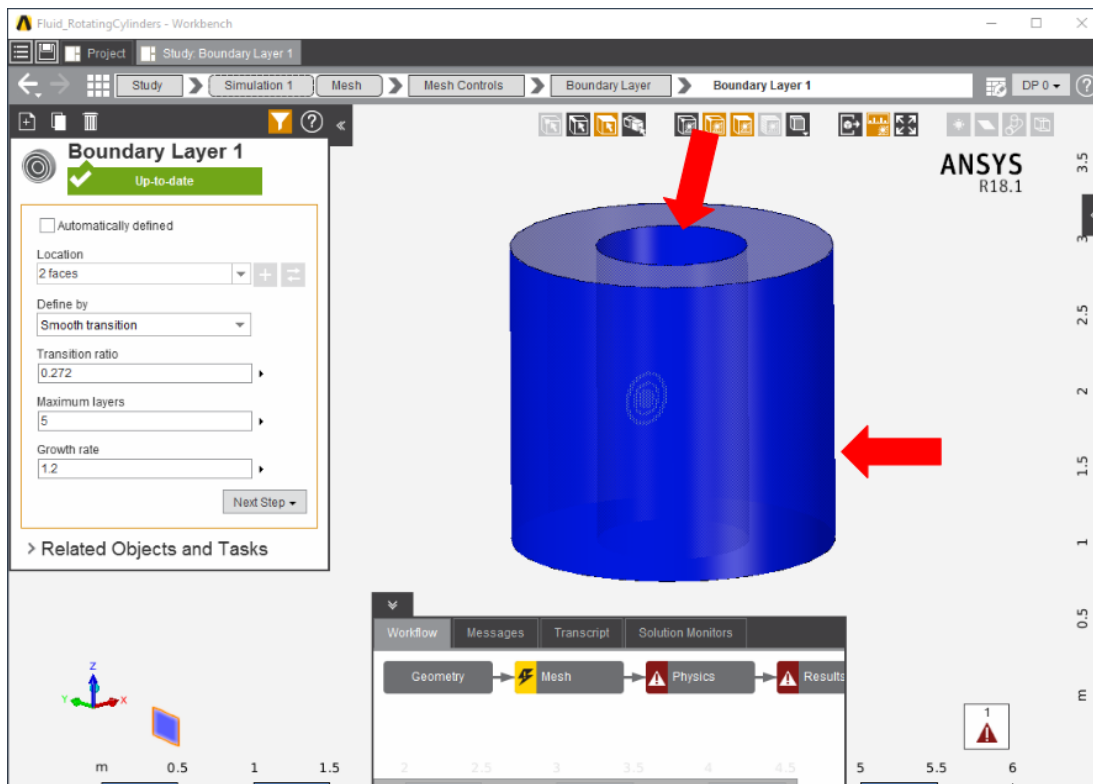
Mesh

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](#)