

ANSYS Taylor-Couette Flow between Cylinders - Geometry

Author(s): Sebastian Vecchi, ANSYS Inc

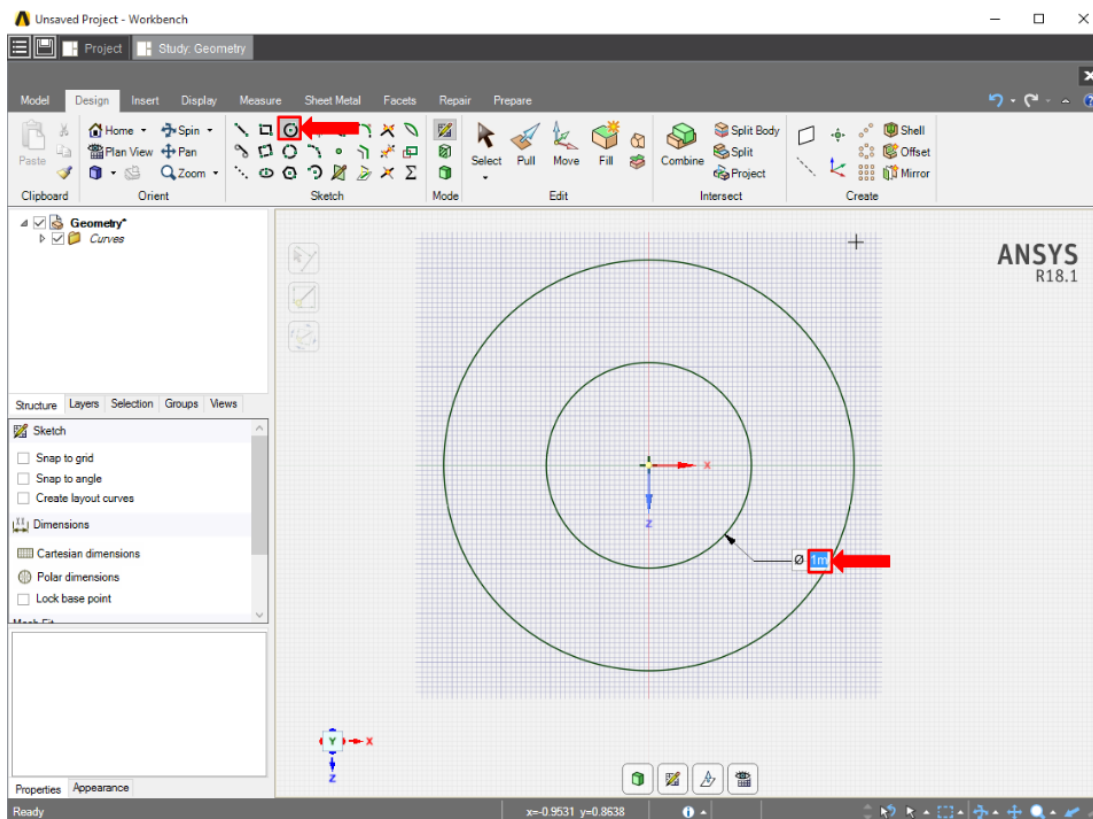
Problem Specification

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

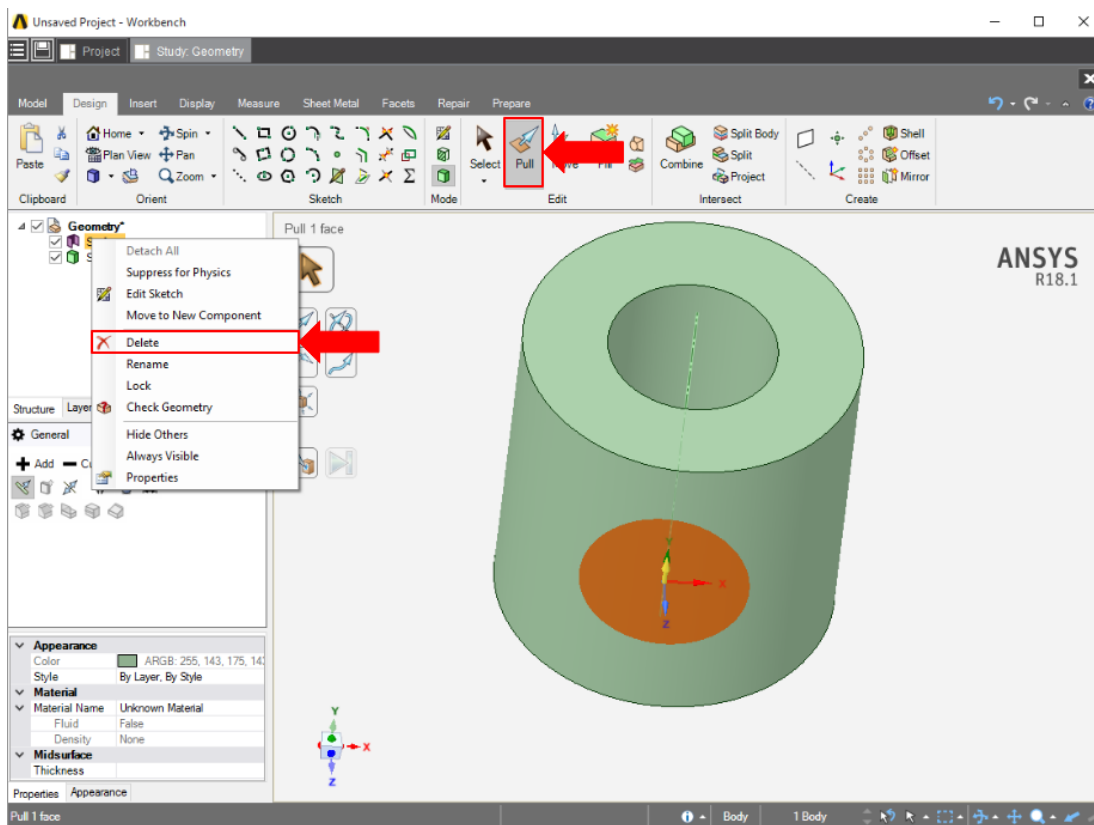
Geometry

Draw Geometry

Select the **Circle** tool and make a circle around the origin with a diameter of 1m. Then, repeat this step with a diameter of 2m.



Press the **Pull** tool in the **Edit** subsection and select the annular surface, then extrude it 2m. Then, right click **Surface** in the **Structure** tree and select **Delete**.



[Go to Step 3: Mesh](#)

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