

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](#)

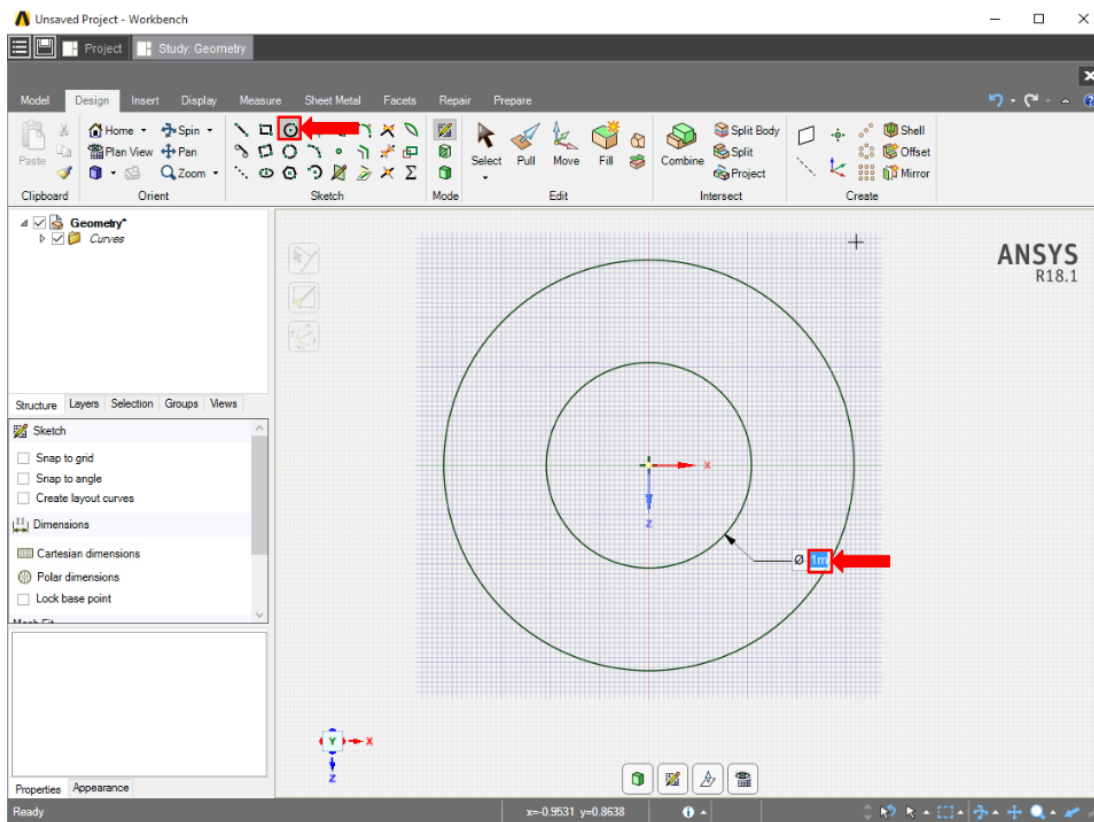
[5. Solution/Results](#)

[6. Verification & Validation](#)

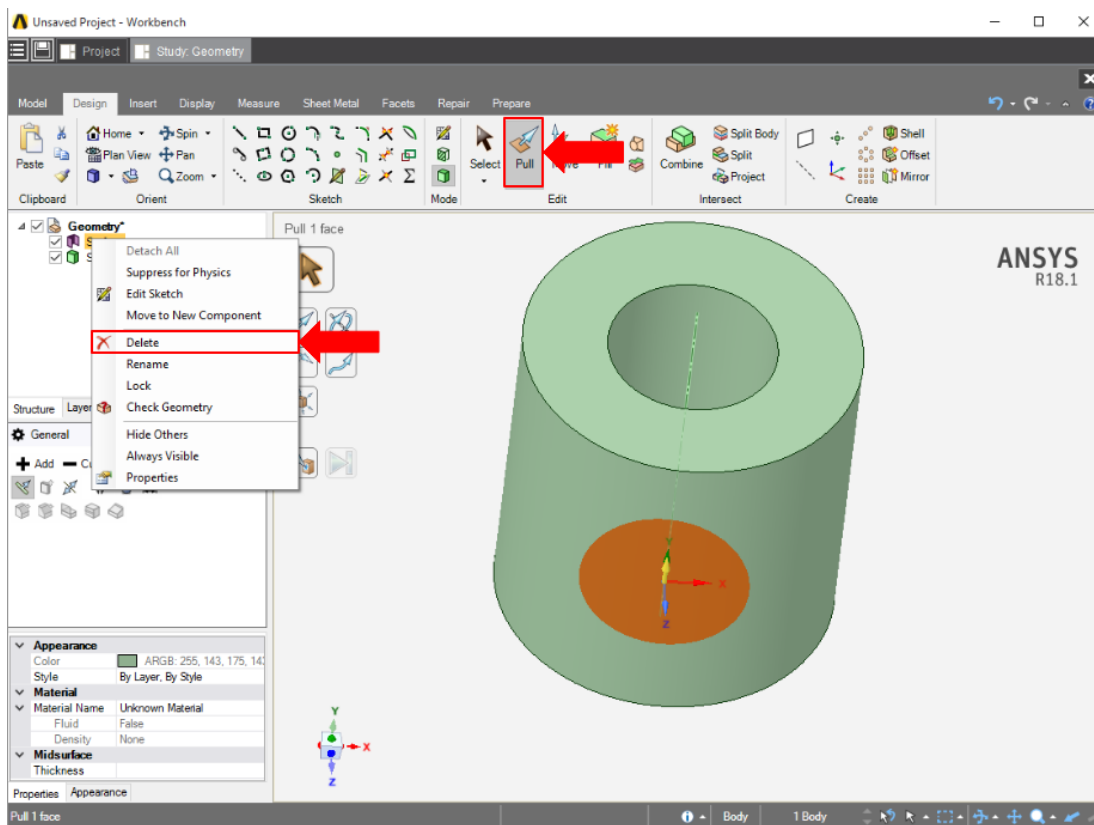
Geometry

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](#)