

Example 3. Modelling of Taylor-Couette Flow for a Newtonian Fluid

3.1. Introduction and Objectives

The viscosity-induced fluid flow that occurs between two parallel walls while one of them is moving and the other remains stationary is called Couette flow. Couette flow can also occur in more complex geometries, for example between two coaxial cylinders. It is important which cylinder is rotating. If the outer, laminar flow is much more stable. The high rotational frequency of the inner cylinder can disturb the laminar fluid flow by the acting centrifugal force. Increasing the rotational speed of the inner cylinder leads to an increase in centrifugal forces that overcome the viscous forces between the liquid particles. This causes the formation of toroidal vortices in the space between two cylinders called Taylor vortices. The fluid flow is then called the Taylor-Couette flow, which will be the object of our interest in this example. The main objectives of this exercise are as follows:

- create 3D geometry of the flow domain from 2D sketch using *Extrude* option,
- illustrate the setup of the simulation with wall in motion – employing the *Rotating Wall* boundary condition,
- visualization of results – contours of axial and radial components of fluid velocity.

3.2. Problem Description

Details of the analysed flow problem are shown in Figure E3-1.

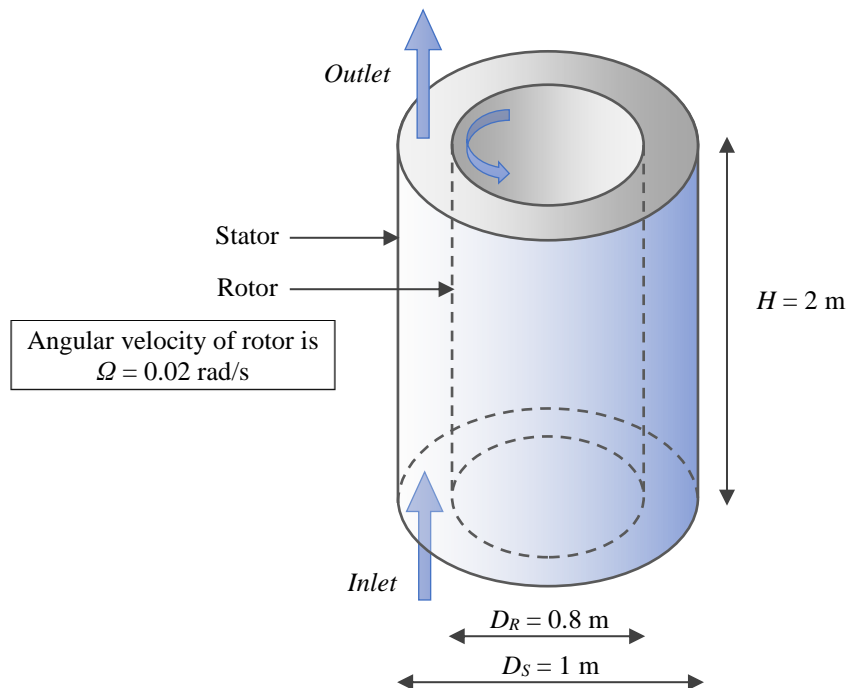
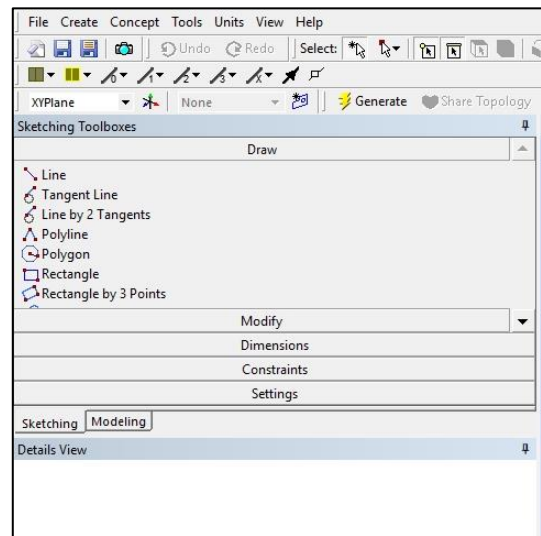


Figure E3-1. Problem description.

3.3. Geometry

Open *Ansys DesignModeler* in order to draw the geometry of the flow domain. In this example, we will first draw a two-dimensional sketch, which will be then extruded to create a three-dimensional object. In this purpose change the *Tree Outline* that is used to display and edit all operations done with 3D objects for the *Sketching Toolboxes* that are used to draw 2D sketches. This can be done by pressing the *Sketching* tab (Fig. E3-2).

Figure E3-2. *Sketching Toolboxes.*



Before we start drawing our geometry, we should setup a proper view. The proper view means a view that is perpendicular to the plane at which we want to draw (an active plane). The default active plane is always *XYPlane* and the default view is always an isometric view. In order to setup a perpendicular view click the *Look At Face/Plane/Sketch* icon that is visible in the main menu (Fig. E3-3). When the view is proper, you can come back to the *Sketching Toolboxes*, where you can find different tabs, such as *Draw*, *Modify*, *Dimensions*, *Constraints* and *Settings* (Fig. E3-3).

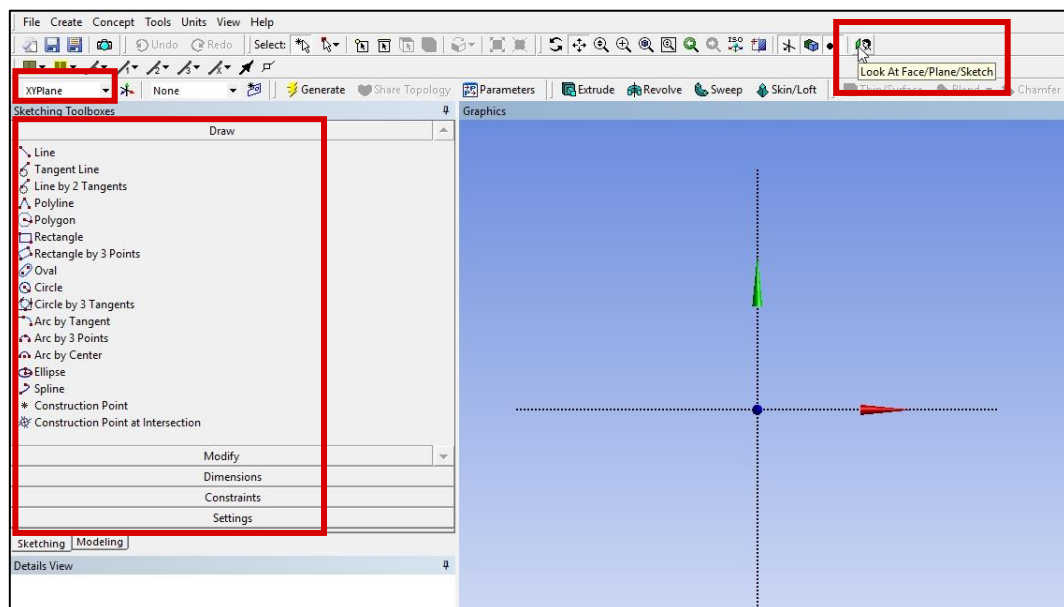


Figure E3-3. Basic view on the *DesignModeler* window with enabled *Sketching* tab.

In the first step of drawing make sure that *Auto Constraints* options are enabled (Fig. E3-4). It is important since they allow new sketch entities to automatically snap to a particular

location or orientation. Choose *Sketching Toolboxes* → *Constraints* → *Auto Constraints* and if they are disabled select both, *Global* and *Cursor* options.

Now go to the *Draw* tab and select *Circle*. Move the cursor to the *Graphics* window and click the point where two axes intersect (the origin of the coordinate system). Make sure you started drawing exactly at this point – this will be indicated by subtle “P” letter near the junction of the axes. Move the cursor away from the centre (but still in the field of *Graphics* window) and click again to finish drawing the geometry (Fig. E3-5a). Do not bother with the dimension of the circle now – we will set all the dimensions in the next step. Now draw the second, larger circle in the same way (Fig. E3-5b).

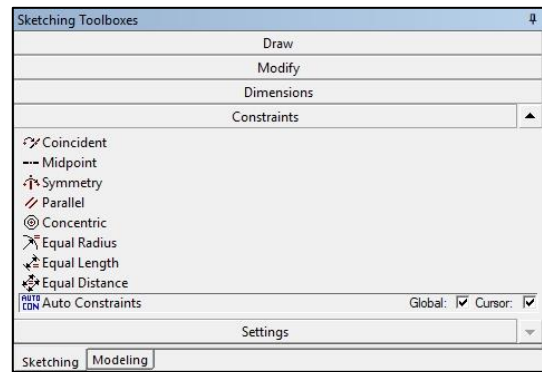


Figure E3-4. *Auto Constraints.*

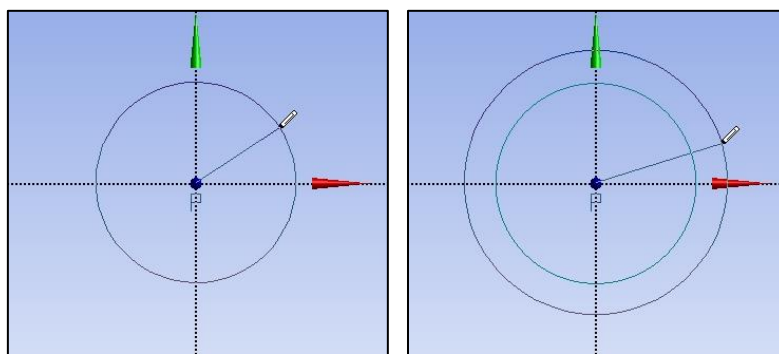


Figure E3-5. The circles drawn in the *Graphics* window.

Now set up the dimensions of the circles. In this purpose open the *Dimensions* tab and select the *General* option that allows you to measure the diameter of the circles. In order to measure the diameter of the larger circle click on the edge of this circle, move the cursor away but still in the field of *Graphics* window and click again. The dimension *D1* was added in the sketch as well as in the *Details view* window. In the same way measure the smaller circle (*D2*). Now in the *Details view* window enter the proper dimensions of the circles – $D1 = 1\text{ m}$ and $D2 = 0.8\text{ m}$ (Fig. E3-6).

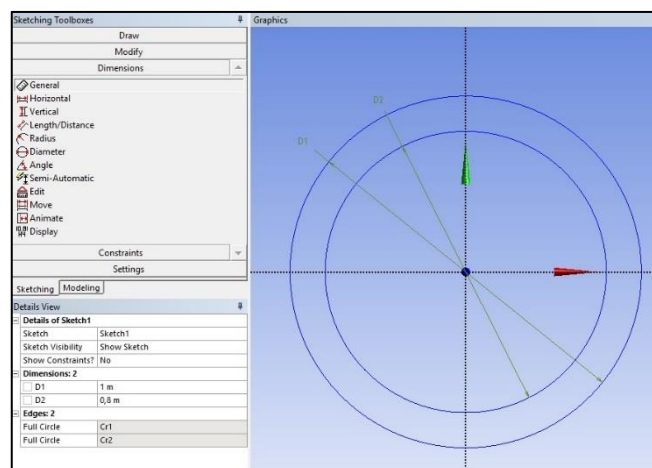


Figure E3-6. The circles dimensioning.

After dimensioning, the sketch is complete and can be converted to 3D geometry. To do this, go back to the *Modeling* tab. You can change the view for the isometric one by clicking on the blue sphere near the coordinate system visible in the right bottom corner (Fig. E3-7a). When you unfold the *XYPlane* in *Tree Outline* and select *Sketch1*, the sketch view is highlighted in yellow in the *Graphics* window. This option is not so important here, but it will be very important for more complex geometries that are drawn in several sketches. When the *Sketch1* is selected choose from the top ribbon the *Extrude* (Fig. E3-7b). In the *Details View* window (Fig. E3-7c), you can set the parameters of *Extrude* option. First click *Apply* to choose the *Sketch1* as a geometry to be extruded. Then enter the *FD1, Depth (>0)* as 2 m. After entering these options in the *Graphics* window you will see a grey wireframe of the geometry to be extruded (Fig. E3-7d). In order to generate the geometry, you have to confirm the operation by clicking *Generate* button. If everything was done correctly, you will see a grey 3D geometry in the *Graphics* window (Fig. E3-7e). You can close *Ansys DesignModeler*.

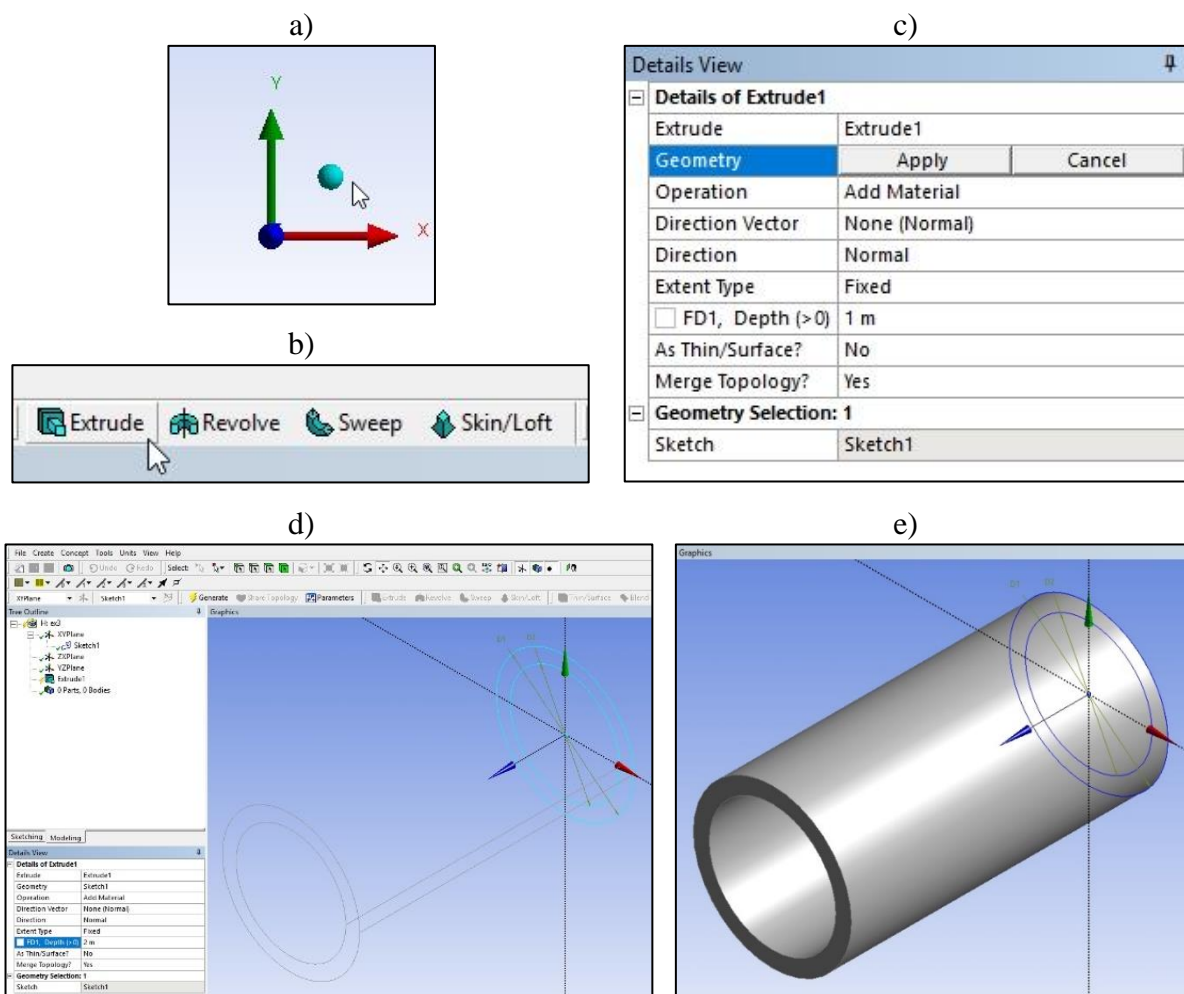


Figure E3-7. Steps of the 3D geometry creating from the 2D sketch.

3.4. Mesh

Open the *Ansys Meshing*, rename the geometry as *Cylinder*, and set up the *Material* as *Fluid*. Next, choose the *Mesh* from the *Tree Outline*. Ensure that in the *Details of "Mesh"* the *Physics Preference* is set as *CFD* and *Solver Preference* as *Fluent*, and confirm by clicking *Generate Mesh*. The grid is now coarse, therefore from the *Top Menu* select the *Mesh* tab and the *Sizing* option in order to improve the grid. *Details of "Sizing"* – *Sizing* window will appear under the *Outline Tree* with several options to set up. Firstly, you need to define the *Geometry* (highlighted with yellow background). In this purpose from the *Graphics Toolbar* select the *Edge* option that allows you to select particular edges directly from the geometry. Press and hold the *Ctrl* button from the keyboard and select four circular edges from the cylindrical geometry of flow domain. When four edges are selected then click once on the *No selection* next to the *Geometry* in the *Details of "Sizing"* – *Sizing* box. *No selection* will change for *Apply/Cancel*. Confirm your choice by *Apply*. Next change the *Type* for the *Number of Divisions* and enter the *Number of Divisions* as 180. Yellow markers appear on the geometry to show 180 equal segments. Regenerate the mesh and examine its *Quality* and *Statistics*. The subsequent steps of mesh generation are presented in Fig. E3-8.

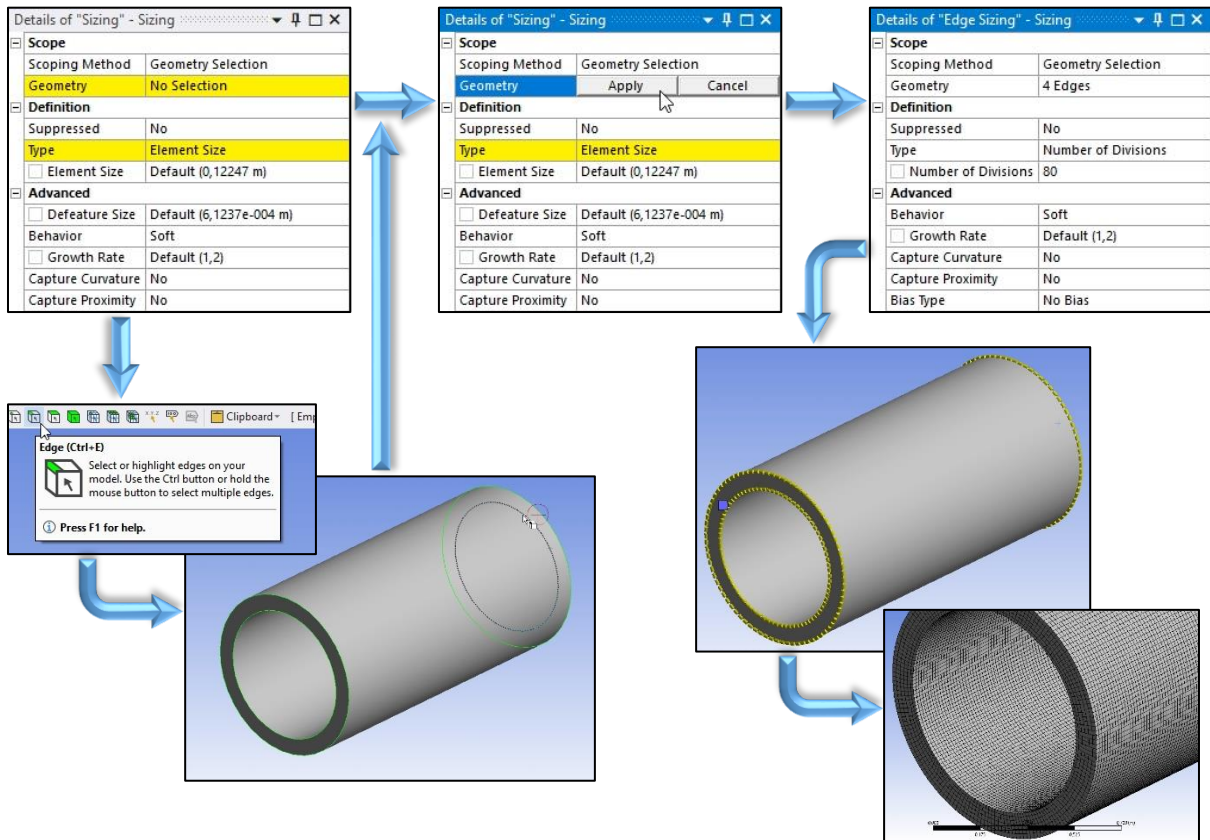


Figure E3-8. Steps of the mesh generation.

In the next step create the *Named Selections* giving them the following names (Fig. E3-9):

1. outer cylindrical surface – *stator*,
2. inner cylindrical surface – *rotor*,
3. bottom annular surface – *inlet*,
4. top annular surface – *outlet*.

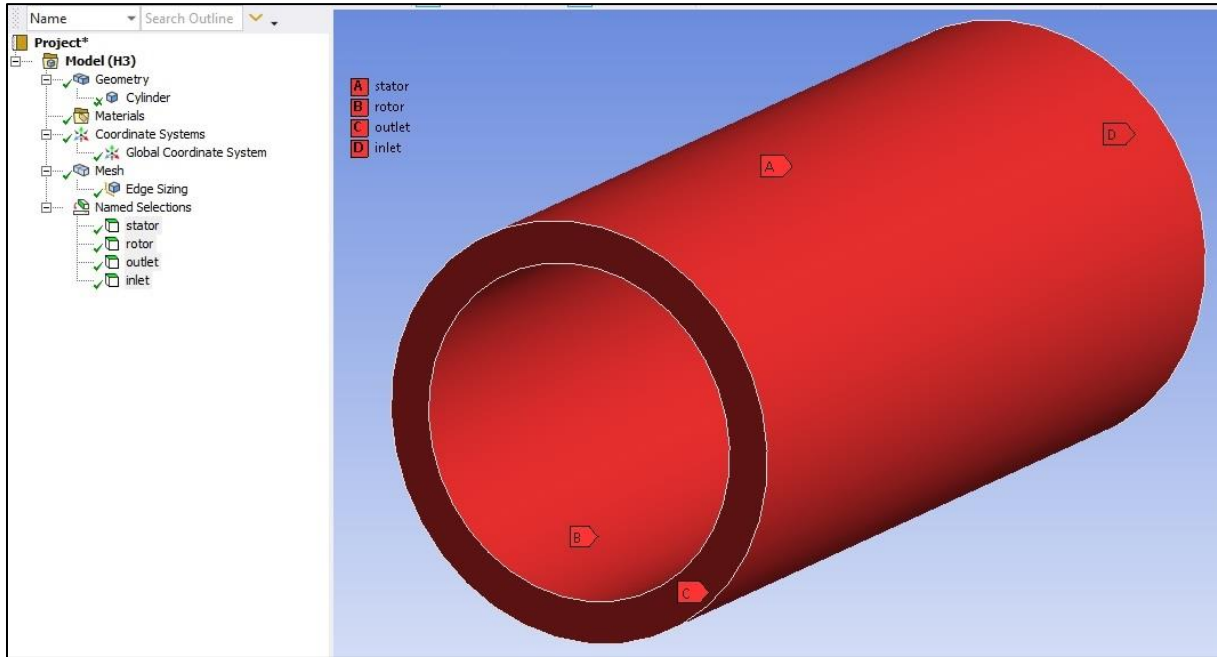


Figure E3-9. *Named Selection*.

Save the project and close *Ansys Meshing*.

3.5. Simulations

In order to perform numerical simulations, open the *Ansys Fluent*. Ensure that *Display Mesh After Reading* option is enabled and the *Solver Processes* is set up as 1. Run down the *Outline View Tree* to edit subsequent parameters and variables:

A. *Setup – General*

1. *Mesh* → *Check*,
2. *Mesh* → *Report Quality*,
3. *Units* → ensure that the unit of the *angular velocity* is set up as *rad/s* (Fig. E3-10),
4. *Solver Type* → *Pressure-Based*,
5. *Solver Velocity Formulation* → *Absolute*,
6. *Time* → *Steady*.

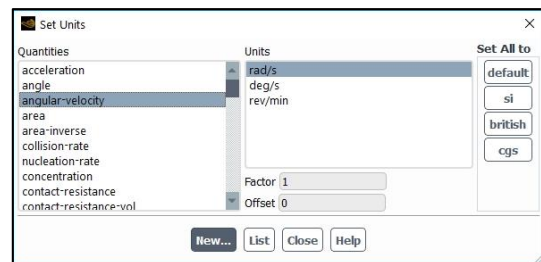


Figure E3-10. *Units settings dialog box*.

B. Setup – Models

Select the *Viscous* → *Laminar* model from the *Models Task Page*.

C. Setup – Materials

Open the *Materials Task Page* and add *water-liquid (h2o<l>)* to the to the *Materials* lists.

D. Setup – Cell Zone Conditions

Open the *Cell Zone Conditions Task Page* and select the *water-liquid* as a process fluid that flows in the annular zone (named as *cylinder*).

E. Setup – Boundary Conditions

Open the *Boundary Conditions Task Page* and firstly set up the boundary conditions as the *Wall* for the *stator* and *rotor*, the *interior* for the *interior-cylinder* zone, the *outflow* for the *outlet* zone and the *velocity-inlet* for the *inlet* zone.

In this case, we will **not define** the velocity of the fluid at the inlet zone, since the rotating inner cylinder is the only factor that drives the flow. Therefore, things we have to change are the settings of the rotor.

In this purpose select the *rotor* from the *Zone* list and press *Edit...* A new *Wall* dialog box will open. Set up the following parameters as (Fig. E3-11):

1. *Wall Motion* → *Moving Wall*,
2. *Motion* → *Relative to Adjacent Cell Zone*,
3. *Motion* → *Rotational*,
4. *Speed [rad/s]* → enter the value *0.02 [rad/s]*
(make sure that before entering value you already changed the type of motion from the default *Translational* for *Rotational*),

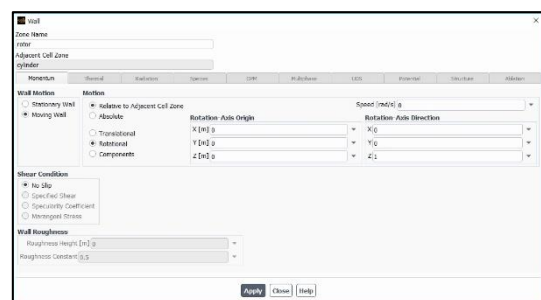


Figure E3-11. *Moving Wall* dialog box.

5. *Shear Condition* → *No Slip*.

After entering the above setup confirm it by clicking *Apply* and close the window. Note that in the *Wall* dialog box there is also a possibility to define the *Rotation-Axis Origin* and *Direction* (Fig. E3-11). If you draw the flow domain geometry exactly like in this tutorial, left these values without change. But if the origin of the rotation axis and/or the rotation axis direction in your geometry are different, you have to enter the proper values here in order to correctly represent the rotation of the computational domain.

F. Solution – Methods

Open the *Solution Methods Task Page* and ensure that the options within this page are set up as: *Scheme* → *Coupled*, *Flux Type* → *Auto Select*, *Gradient* → *Least Squares Cell Based*, *Pressure* → *Second Order*, *Momentum* → *Second Order Upwind*, and other options are disabled.

G. Solution – Monitors

Open the *Residual Monitors* dialog box, make sure that all equations are monitored and their convergence is checked. Then set all *Absolute Criteria* as $1e-7$.

H. Solution – Initialization

Open the *Solution Initialization Task Page* and initialize the calculation selecting the *Standard Initialization* with enabled *Compute from all-zones* option.

I. Solution – Run Calculation

Open the *Run Calculation Task Page* and examine your setup selecting *Check Case...* You should see the recommendation about flow specification at the inlet boundary. Nevertheless, as it was explained, left the default 0 [m/s] value. If there are no other recommendations, you can start the calculation by entering 350 iterations. During the calculations in the main window you will observe the *Scaled Residuals* (Fig. E3-12) that do not fall as much as in Examples 1 and 2 without moving parts.

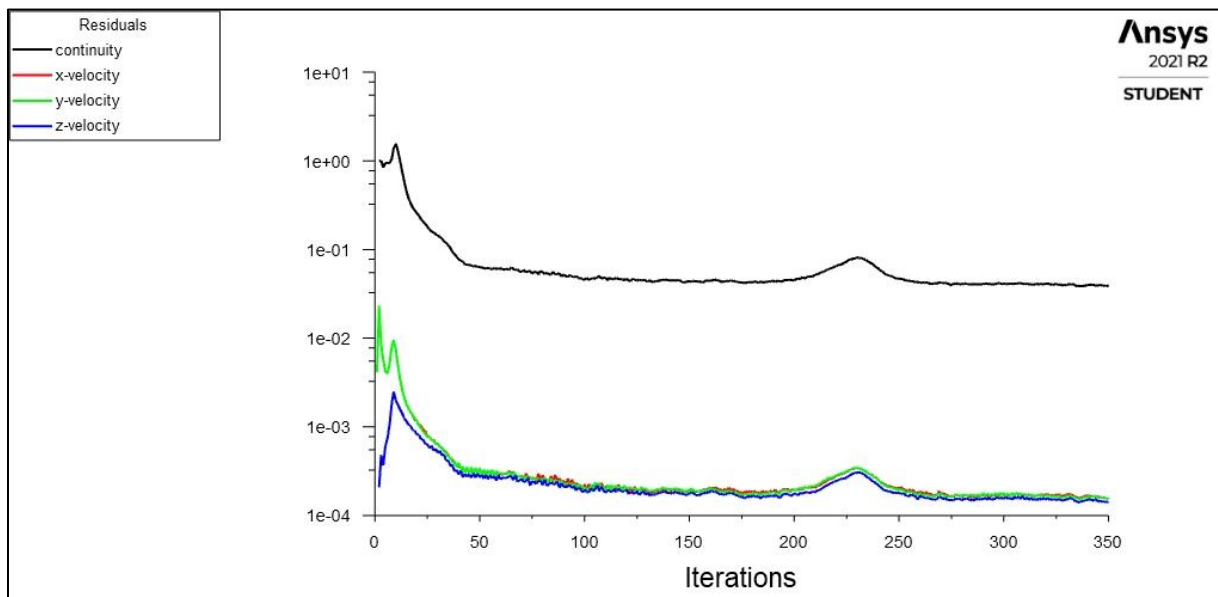


Figure E3-12. Residuals for the converged solution.

3.6. Results

In this example results will be presented as a contours of the axial and radial fluid velocity components on the rotor surface and in the gap between the inner and outer cylinders (rotor and stator). The rotor surface is available on the *Surfaces* lists of *Contours* dialog box, but surface representing the gap between two cylinders (a vertical mid-plane of the flow domain) have to be created. To do this examine section 1.6 (Example 1).

To create contours of the axial components of fluid velocity open the *Contours* dialog box and enter the following settings (E3-13):

1. *Contours of* → *Velocity...* → *Axial Velocity*,
2. *Surfaces* → select *rotor* and new plane that you created, here it is *plane-4*,
3. *Coloring* → *Smooth*,
4. Options such as *Filled*, *Node Values*, *Global Range* and *Auto Range* should be enabled.

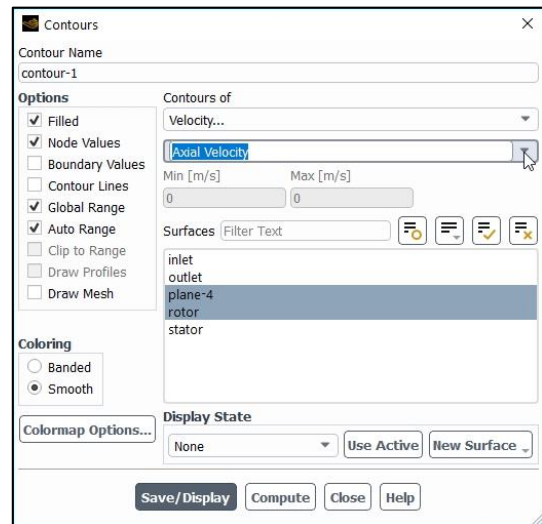


Figure E3-13. Contours dialog box.

Click *Save/Display* button. After checking the results create similar one by choosing the *Radial Velocity* from the dropdown list. The results are presented in Figure E3-14.

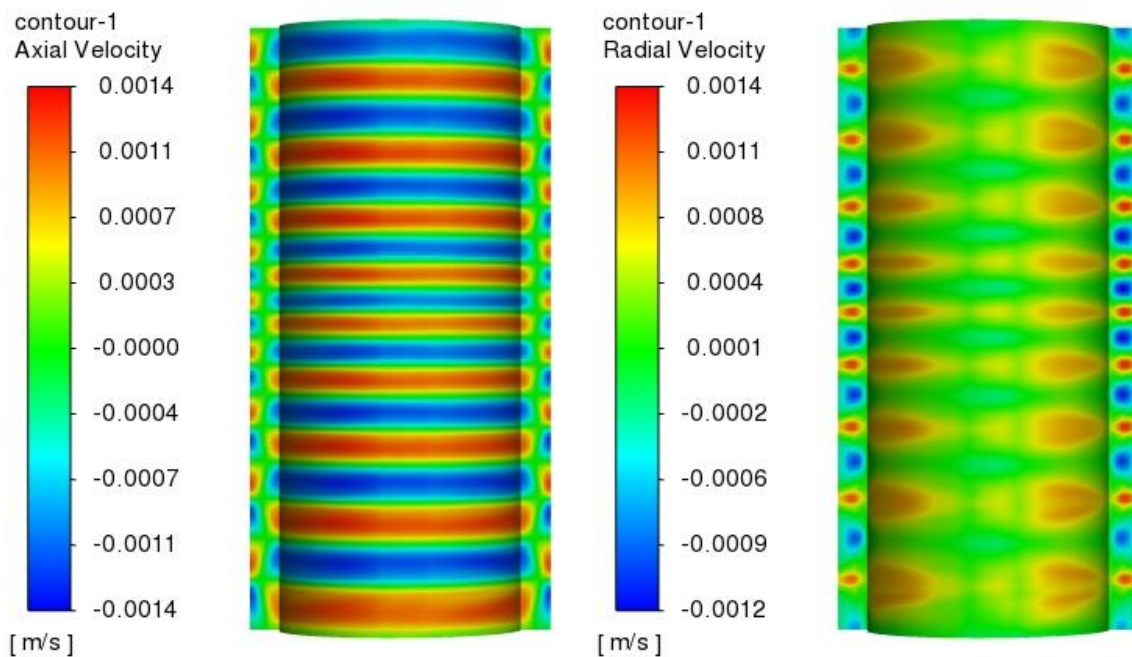


Figure E3-14. Contours of axial and radial velocity components.