

Example 4. Modelling of Taylor-Couette Flow for a non-Newtonian Fluid

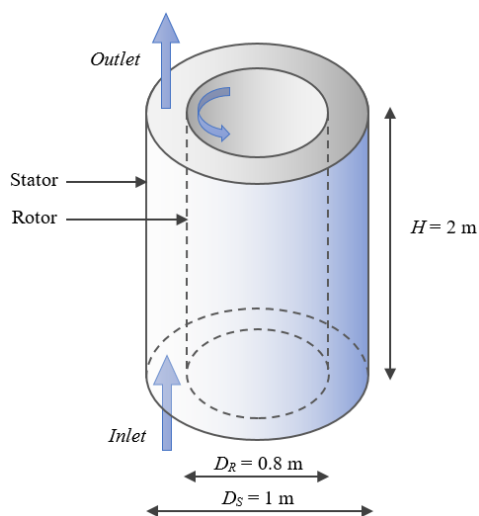
4.1. Introduction and Objectives

The previous examples concerned simulations for water, a liquid that obeys Newton's law of friction. The viscosity of the Newtonian fluid does not depend on the shear rate, i.e. under certain pressure and temperature conditions it has a constant value. There is a group of fluids for which the relationship of shear stress as a function of shear rate cannot be described by a simple linear Newton equation. The behaviour of these fluids differs significantly from Newtonian fluids, therefore they are generally referred to as non-Newtonian fluids. Non-Newtonian fluids include systems with very different rheological properties and are often used as working media in industry. They do not obey Newton's law of friction and, unlike Newtonian fluids, their viscosity strongly depends on the shear rate. The complex rheological properties of non-Newtonian fluids make the flow task more complicated and difficult to analyse. For such flow tasks, the system of equations has to be supplemented with an appropriate rheological model describing the fluid viscosity under certain conditions of pressure, temperature and shear rate. In this example again we will consider Taylor-Couette flow, however for a complex non-Newtonian fluid. The main objectives are as follows:

- define a new material that is not implemented into the Fluent database of materials,
- illustrate the setup of the non-Newtonian fluid flow simulation,
- visualization of results – contours of axial and radial components of fluid velocity and contours of viscosity.

4.2. Problem Description

Details of the analysed flow problem are shown in Figure E4-1.



- I. Two levels of angular velocity are tested:
 $\Omega_1 = 0.02 \text{ rad/s}$ and $\Omega_2 = 7.5 \text{ rad/s}$,
- II. Tested fluid: 0.2% Carbopol Ultrez 30 with pH = 5.0 that can be described with Herschel-Bulkley model with the following values of rheological parameters:
 - Consistency Index, $k = 10.7 \text{ Pas}^n$
 - Power-law Index, $n = 0.459$
 - Yield Stress Threshold, $\tau_y = 19.6 \text{ Pa}$
 - Critical Shear Rate, $\dot{\gamma} = 3.761 \text{ s}^{-1}$

Figure E4-1. Problem description.

4.3. Geometry and Mesh

Geometry and mesh representing the flow domain in this example are the same like in the Example 3. Therefore, to prepare them follow the steps described in the Example 3. You can also use the previously generated geometry/mesh in this Example.

4.4. Simulations

Most of the settings in this example are the same like in the previous one, therefore they will not be repeated here. In this part of the tutorial only the differences between settings within these two examples are described. The user is therefore obligated to verify the remaining settings with the previous instructions.

A. Setup – Materials

The present simulation will be carried out for a new material that is not available in the *Fluent Database*. Therefore, the user has to introduce it into the software. According to the *Problem Description*, a new material is a 0.2% Carbopol Ultrez 30 with pH = 5.0 that is a highly non-Newtonian microgel which can be described with the Herschel-Bulkley model. In order to add this liquid into the *Ansys Fluent* open the *Materials Task Page* and press *Create/Edit...* button. In the *Create/Edit Materials* dialog box (Fig. E4-2) enter the name of a new material as *carbopol* and remove the *Chemical Formula* (left this place blank). Next from the *Viscosity* dropdown list select the *herschel-bulkley* model.

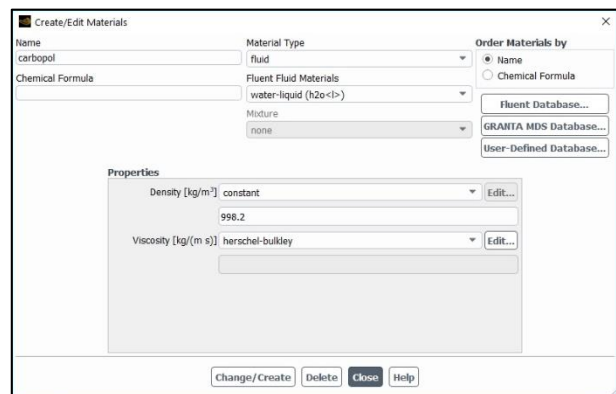


Figure E4-2. Entering of a non-Newtonian material to the *Materials* list.

This will open a new dialog box (Fig. E4-3) where you have to enter values of the Herschel-Bulkley model presented in the description of the problem. After entering values of the rheological parameters click *OK* to close Herschel-Bulkley dialog box. Then in the *Create/Edit Materials* dialog box press the *Change/Create* button.

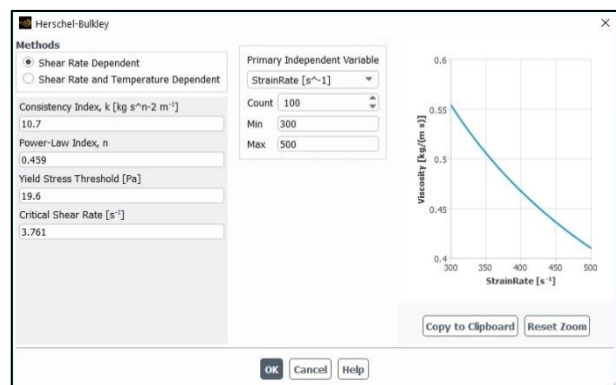


Figure E4-3. Entering values of the rheological parameters of a non-Newtonian liquid.

A new dialog box will open with the following question: “*Change/Create mixture and Overwrite (name of previous material)?*” (Fig. E4-4). If you choose *Yes*, then a new material will appear on the *Materials* list but at the same time the previous material which was there will be removed. If you choose *No*, then both materials, old and a new one will be kept and visible on the *Materials* list. In this exercise select “*No*”. You can close the *Create/Edit Materials* dialog box.



Figure E4-4. The last stage of creating a new material in the *Ansys Fluent* code.

B. Setup – Cell Zone Conditions

Open the *Cell Zone Conditions Task Page* and select the new created material – *carbopol*, as a process fluid that flows in the annular zone.

Remember about other settings within the *Ansys Fluent* connected with the general setup, models, boundary conditions, methods and monitors of the solution, initialization. When all the settings will be ready run the calculation entering 350 iterations. As the iterative process will be completed check the results.

4.5. Results

The angular speed of the rotor in the first simulation was equal to $\Omega_1 = 0.02$ rad/s. You should obtain contours of the axial and radial velocity similar to those presented in Fig. E4-5.

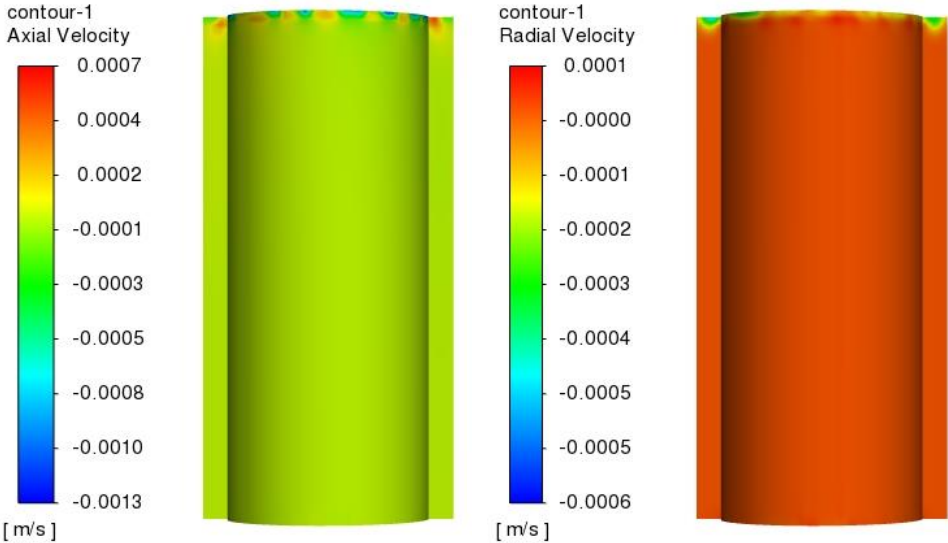


Figure E4-5. Contours of axial and radial velocity components for a non-Newtonian fluid, $\Omega_1 = 0.02$ rad/s.

As you can observe from the analysis of Fig. E4-5 in the flow domain there is no Taylor vortices which were observed in the previous example where as a process fluid the water was chosen. In contrast to the water, the Carbopol microgel is a non-Newtonian fluid that is characterized by a very high value of apparent viscosity, especially under the low shear forces. Recall from the *Fluid mechanics* course that viscosity describes the internal friction of a moving fluid. In other words, it is a measure of a fluid's resistance to flow. In our example the rotating inner cylinder is the only one factor that induces the flow. The conclusion is that the entered angular velocity of the rotor is too low to withstand the viscous forces in Carbopol. Therefore, come back to the *boundary condition* and enter a new value of the angular velocity of the inner cylinder as $\Omega_2 = 7.5$ rad/s. **Do not reinitialize the process.** When you change value of the rotor angular velocity go directly to the *Run Calculation* tab and run the calculation again entering another 350 iterations. A warning will appear that the settings have changed. Confirm this information choosing “*Use settings changes for current calculation only*” and clicking *OK*. The program will continue calculations starting from the previous result. If you reinitialize the process, the old results will be deleted and the simulations will start all over again.

Results of the new simulation performed for the increased value of the angular velocity of the rotor should reveal Taylor vortices in the Taylor-Couette flow occurring between two coaxial cylinders, as it is shown in Fig. E4-6. Nevertheless, since the rheological properties of the tested Carbopol microgel are completely different than for water, also the vortices in this flow task have a different shape.

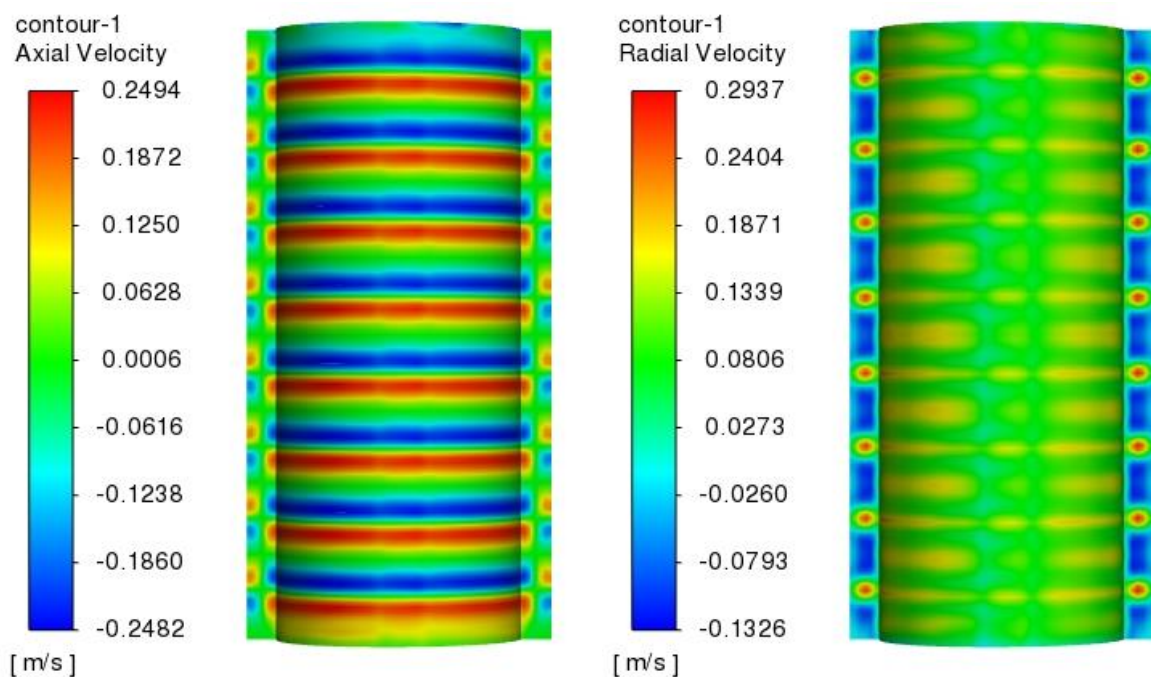


Figure E4-6. Contours of axial and radial velocity components for a non-Newtonian fluid, $\Omega_1 = 7.5$ rad/s.