### Example 2. Modelling of Turbulent Flow of Newtonian Fluid in Pipe

## 2.1. Introduction and Objectives

In the previous example we said that internal flows are widespread in all industries and we analysed a simple laminar flow through a pipe. However, most flows encountered in engineering practice are turbulent. Turbulent flow, instead of laminar one, does not flow in parallel layers but is characterized by random, irregular (chaotic) and rapid movement of particles of the fluid and, consequently, a significant variation of the flow properties in space and time. One of the advantage of the turbulent flow is highly efficient mixing of the fluid. On the other hand, it is much more complex than the laminar flow, thus the equations derived for the laminar flow are not sufficient to describe its random nature. In the case of the turbulent flow problem, the system of equations is supplemented with so-called turbulence model. In this example we will consider the turbulent flow ( $Re = 10\,000$ ) of a Newtonian fluid (water) through a pipe with the same dimensions as in exercise 1.

The main objectives of this exercise are as follows:

- simulations for the existing geometry of the computational domain and numerical mesh,
- illustrate the setup of the turbulent fluid flow simulation,
- visualization of results contours and profiles of fluid velocity, contours of properties connected with turbulence.

# 2.2. Problem Description

Details of the analysed flow problem are shown in Figure E2-1.



Figure E2-1. Problem description.

## 2.3. Geometry and Mesh

Geometry and mesh representing the flow domain in this example are the same like in the previous example. Therefore, to prepare them follow the steps described in the following

sections of Example 1: *1.3. Geometry* and *1.4. Mesh.* If you already did the first example you can use now the mesh that was generated earlier, for example by creating the connectors between the *Mesh* cell from the module of the previous example and the *Setup* cell from the module of the present example, as it is shown in the Fig. E2-2.



Figure E2-2. Connectors between *Mesh* cell of Example 1 and *Setup* cell of Example 2.

### 2.4. Simulations

In general, the flow task presented in this example is very similar to that analysed in the Example 1. The only difference is that previously we examined the laminar flow, while now we have the turbulent flow of the water in the pipe since the Reynolds number was increased from Re = 500 to  $Re = 10\ 000$ . From this reason most of the settings within the *Ansys Fluent* will not change and therefore they will not be repeated in details within this section. The user is obligated to check carefully all the settings from this example by comparing them to the settings of the Exercise 1. All the new settings will be described with details.

In order to perform numerical simulations, open the *Ansys Fluent*. Ensure that the *Double Precision* and *Display Mesh After Reading* are 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  $\rightarrow$  Check,
  - 2. Mesh  $\rightarrow$  Report Quality,
  - 3. Solver Type  $\rightarrow$  Pressure-Based,
  - 4. Solver Velocity Formulation  $\rightarrow$  Absolute,
  - 5. *Time*  $\rightarrow$  *Steady*.

### B. Setup – Models

Open the *Models Task Page* by double-click the *Models* in the *Outline View Tree*. Next double-click on *Viscous* model from the list of available models. A new dialog box will open,

that call *Viscous Model*. For turbulent flow analysis there are several models that approximate the solution. In this example we will use the k- $\varepsilon$ **turbulence model**. Select the *k-epsilon (2 eqn)* model from the model list in *Viscous Model* dialog box. The dialog box will expand showing a new options characteristic for chosen turbulence model (Fig. E2-3). Use the default *Standard* from the *k-epsilon Model* group and choose *Enhanced Wall Treatment* from the *Near-Wall Treatment*. Do not change the values of *Model Constants*. Click *OK* to accept the model settings and close the *Viscous Model* dialog box.



Figure E2-3. Viscous Model dialog box with enabled k-epsilon model.

### 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 *pipe* zone.

#### E. Setup – Boundary Conditions

Open the *Boundary Conditions Task Page* and set up the boundary conditions as the *Wall* for the outer cylindrical *wall* zone, the *interior* for the *interior-pipe* zone, the *pressure-outlet* for the *outlet* zone and the *velocity-inlet* for the *inlet* zone. For the last boundary condition enter the *Velocity Magnitude* [m/s] as w = 0.0201 [m/s].

### F. Solution – Methods

Open the Solution Methods Task Page and ensure that the options within this page are set up the same as in the previous example, i.e.: Scheme  $\rightarrow$  Coupled, Flux Type  $\rightarrow$  Auto Select, Gradient  $\rightarrow$  Least Squares Cell Based, Pressure  $\rightarrow$  Second Order, Momentum  $\rightarrow$  Second Order Upwind, and other options are disabled. Note that comparing to the previous example, here also the *Turbulent Kinetic Energy*, k, and *Turbulent Dissipation Rate*,  $\varepsilon$ , are calculated. For both these parameters select from the dropdown list the *Second Order Upwind* scheme of spatial discretization.

# 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-13. Note that except *continuity* and *velocity components* now we can find here also  $\mathbf{k}$  and  $\varepsilon$  – parameters that come from the turbulence model.

### H. Solution – Initialization

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

# I. Solution – Run Calculation

Open the *Run Calculation Task Page* and examine your setup selecting *Check Case*... You should see the *Case Check* dialog box with recommendation to consider the *realizable k-epsilon turbulence model* instead of *standard* one (Fig. E2-4). Nevertheless, the analysed flow problem is quite simple, without complex structures, rotation, separation or recirculation. Therefore, we do not apply this recommendation. If there are no other recommendations, you can click *Close* in the dialog box and start the calculation by entering 200 iterations. During the calculations in the main window you will see the *Scaled Residuals* for each of the six equation parameters. They are presented in Fig. E2-5.

(vie	esh 🛛 🔊	Models	Boundaries and Cell Zones Materials Solve						
itoma	tic Implementa	ation							
Apply	Recommendatio	on							
V	Consider realiz (Models: Edit V	able k-eps /iscous)	silon in lieu of the standard k-	epsilon turbulence	model. ?				

Figure E2-4. Case Check dialog box with recommendation.



Figure E2-5. Residuals for the converged solution.

### 2.5. Results

In this example results will be presented in the same way like previously, as a contours and profiles of the fluid velocity. However, here also contours of the turbulent kinetic energy and turbulent dissipation rate will be presented.

Examine section 1.6 in order to create a mid-plane of the pipe and lines representing the diameter of the pipe at three locations: 0.5 m, 2.5 m and 4.5 m starting from the pipe inlet. The velocity magnitude contours and profiles results are presented in Figs. E2-6 and E2-7, respectively. You can compare these results with the results from the Example 1.

Contours of the turbulent kinetic energy and turbulent dissipation rate were created in the same way as velocity contours with such difference that in the *Contours* dialog box the *Turbulence*... was selected instead of the *Velocity*... (Fig. E2-8). Next, from the dropdown list first the *Turbulent Kinetic Energy* (k), and then the *Turbulent Dissipation Rate* (*epsilon*) were selected. They are presented in a mid-plane Figure E2-9.



Figure E2-6. Contours of fluid velocity magnitude in the mid-plane, at the inflow and outflow surfaces.



Figure E2-7. Profiles of velocity magnitude along the pipe diameter in three locations.

contour-1							
Options	Contours of						
✓ Filled	Turbulence	Ŧ					
✓ Node Values	Turbulent Kinetic Energy (k)						
Boundary values	Min [m/s]	Max [m/s]					
	0	0.02298999					
<ul> <li>✓ Global Range</li> <li>✓ Auto Range</li> </ul>	Surfaces Filter Text						
Clip to Range Draw Profiles Draw Mesh	inlet line-0.5 line-2.5 line-4.5						
	mid-plane						
Coloring	outlet wall						
<ul> <li>Banded</li> <li>Smooth</li> </ul>							
Colorman Ontions	Display State						
colorniap options	None Vise Active New Surface						

Figure E2-8. Contours dialog box set up for the contours of the turbulent kinetic energy.

conto Turbu	ur-1 Ilent Dis	sipation Rat	e (Epsilon) [	m^2/s*3]							
2.7	3 e-09	5.51 e-08	1.08e-07	1.60e-07	2.12e-07	2.65e-07	3.17e-07	3.70e-07	4.22e-07	4.74e-07	5.27e-07

Figure E2-9. Contours of turbulent kinetic energy and turbulent dissipation rate in the pipe mid-plane.