

Example 1. Modelling of Laminar Flow of Newtonian Fluid in a Pipe

1.1. Introduction and Objectives

A flow that is completely bounded by a solid surface is called an internal flow. Internal flows are widespread in all industries as they involve many important and practical flows such as flow through pipelines and accompanying fittings. Depending on value of the Reynolds number, internal flows can be laminar, transitional or turbulent. In this simple example we will consider the laminar flow ($Re = 500$) of a Newtonian fluid (water) through a pipe with a smooth surface. The main objectives of this exercise are as follows:

- draw 3D geometry of the flow domain using *Primitives* options,
- generate a structured mesh,
- identify and create *Named Selection* – individual parts of flow domain to which different materials or boundary conditions will be assign,
- illustrate the setup of the laminar fluid flow simulation,
- visualization of results – contours and profiles of fluid velocity.

1.2. Problem Description

Details of the analysed flow problem are shown in Figure E1-1.

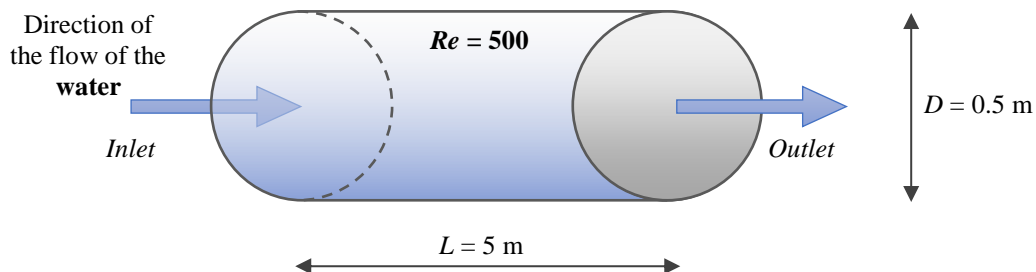


Figure E1-1. Problem description.

1.3. Simulations

Open the *Ansys Fluent* software by a double-click the *Setup* in the *Fluent* module. During the first start of *Ansys Fluent*, the *Fluent Launcher* (Fig. E1-2) will open, where the user can check and set a several start-up options. Enter the *Double Precision*, *Display Mesh After Reading* and *Solver Processes* equal to 1. Then click *Start* in order to launch *Ansys Fluent*. Import the mesh available on adylak.zut.edu.pl.

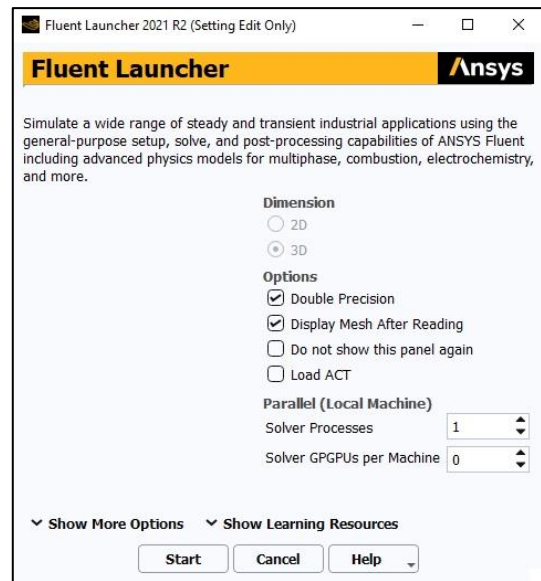


Figure E1-2. Fluent Launcher.

The *Ansys Fluent* window (Fig. E1-3) is segmented into several menus. On the top ribbon we can find various tabs, e.g. *File*, *Domain*, *Physics*, *Solution*, etc., where all adjusting properties are collected. Below the top ribbon there is an *Outline View* that include each step of setting up the simulation, a *Task Page* where we can enter the variables and properties available in each step of the *Outline Tree*, and the main window where we can observe the domain, simulation progress or results. There is also a *Console* under the main window – a command line that shows the progress of simulation or errors. Here, the user can also enter specific commands to perform particular tasks.

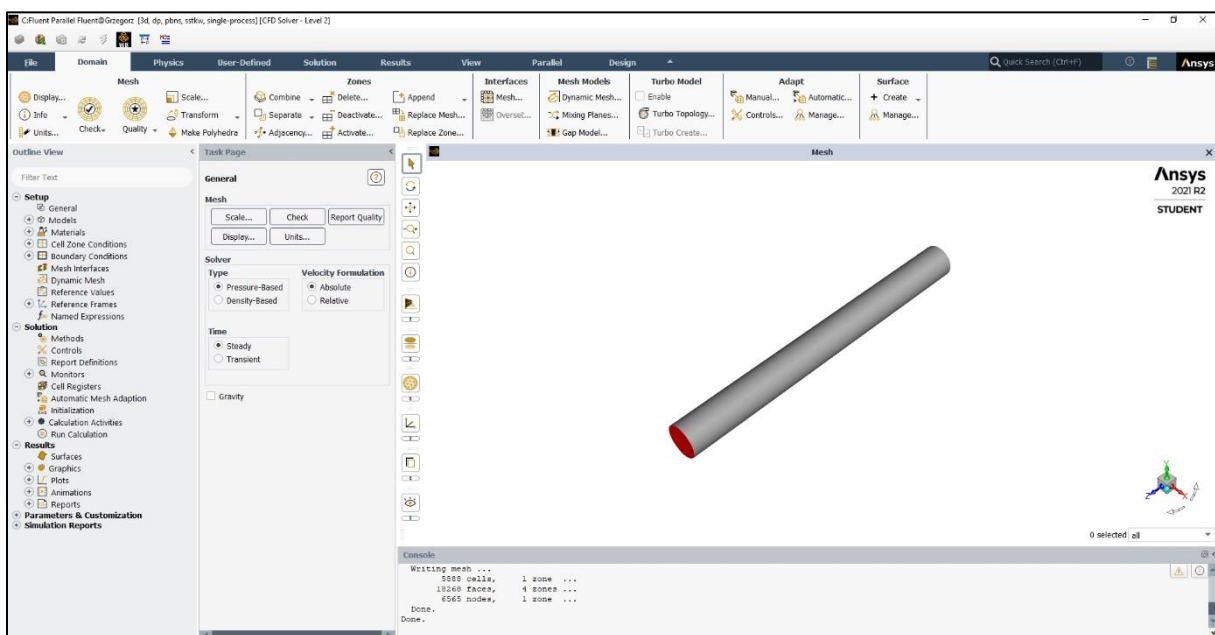


Figure E1-3. Ansys Fluent layout.

Operating the *Ansys Fluent* will mainly consist of running down the tree in *Outline View* and editing parameters and variables in each respective *Task Page*.

A. Setup – General

From the *General Task Page* (Fig. E1-4) choose the following options:

1. **Mesh** → **Check** – program will report the results of the mesh check in the console. Make sure that the minimum volume is not negative as the calculation cannot begin in this case.
2. **Mesh** → **Report Quality** – program will report the results of the mesh quality in the console. Values of cell orthogonality can vary between 0 and 1, where 0 correspond to a very poor cell quality. In general, the minimum orthogonality should not be below 0.01 with the average value significantly larger.

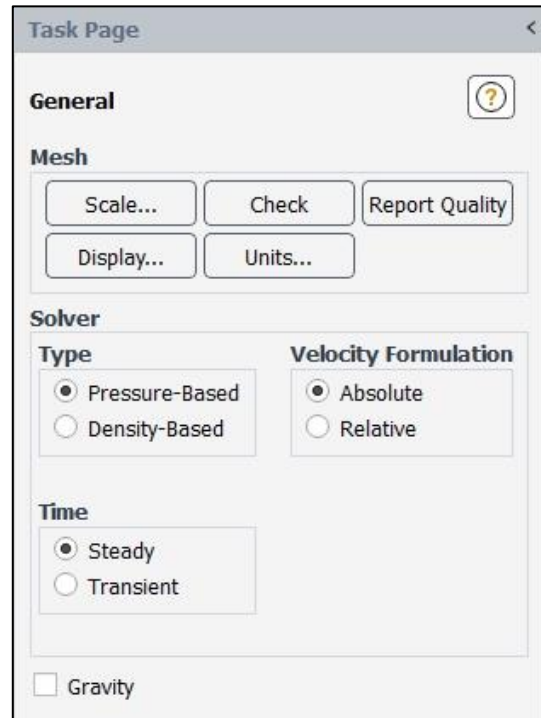


Figure E1-4. *General Task Page.*

3. **Solver Type** → **Pressure-Based** – proper for low-speed incompressible to high-speed compressible flows.
4. **Solver Velocity Formulation** → **Absolute** – proper for cases where majority of the domain is non-rotational.
5. **Time** → **Steady** – for a steady-state solution.

B. Setup – Models

The purpose of this flow task is to simulate the laminar flow of a liquid, therefore only **Viscous** → **Laminar** should be selected from the list of available models. Open the *Models Task Page* (Fig. E1-5) by double-click the *Models* in the *Outline View Tree*. Make sure that the *Laminar* model is selected within the *Viscous* tab.

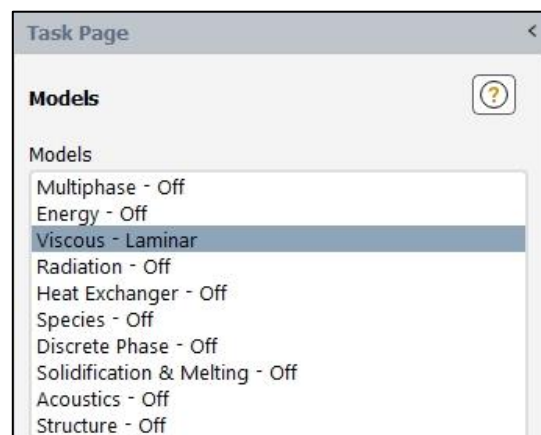


Figure E1-5. *Models Task Page.*

C. Setup – Materials

Open the *Materials Task Page* (Fig. E1-6) by double-click the *Materials* in the *Outline View Tree*. Default materials are *air* as a fluid and *aluminum* as solid. The presented simulation is performed for the water therefore it is necessary to create this material. In this purpose click *Create/Edit...* button visible at the bottom of the *Materials Task Page* that launch a new window – *Create/Edit Materials* (Fig. E1-7a). Water is a material that is available in the *Ansys Fluent Database*. Click on the *Fluent Database...* on the right. In the new window (*Fluent Database Materials*, Fig. E1-7b) find in the list the ***water-liquid (h2o<l>)***. Select this material and confirm with *Copy*. Material was automatically added to the *Materials* lists on the *Materials Task Page*. You can close both *Fluent Database Materials* and *Create/Edit Materials* windows.

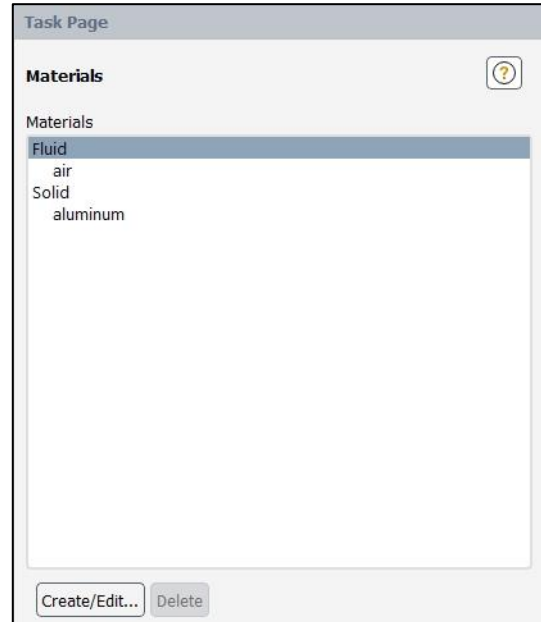


Figure E1-6. *Materials Task Page*.

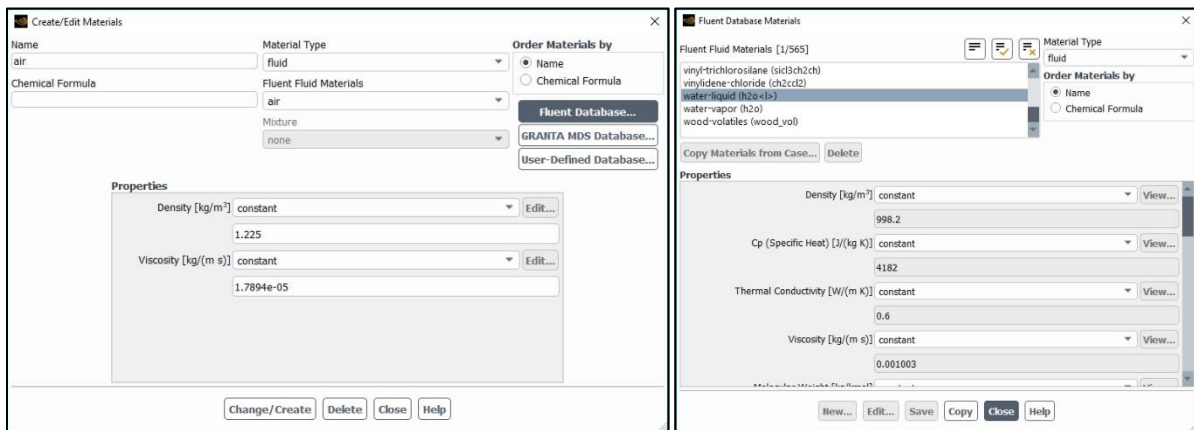


Figure E1-7. Adding a new material from the *Fluent Database*.

D. Setup – Cell Zone Conditions

Open the *Cell Zone Conditions Task Page* (Fig. E1-8a) by double-click the *Cell Zone Conditions* in the *Outline View Tree*. From the *Zone* list select *pipe* and click *Edit...*. In a new *Fluid* window (Fig. E1-8b) the default material is the air. Change the *Material Name* for the ***water-liquid*** by selecting it from the dropdown list and confirm with *Apply* button. The rest setting left without change and close the *Fluid* window.

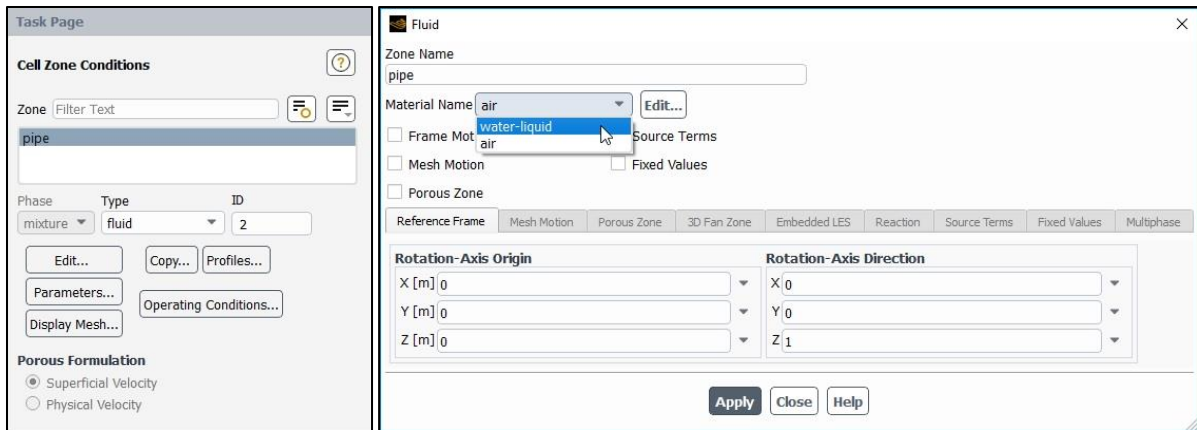


Figure E1-8. Setup of the *Cell Zone Conditions*.

E. Setup – Boundary Conditions

Open the *Boundary Conditions Task Page* (Fig. E1-9a) by double-click the *Boundary Conditions* in the *Outline View Tree*. On the *Zone* list there are sections that were created in the *Ansys Meshing*, i.e. *inlet*, *outlet* and *wall*. Also, the *interior-pipe* zone was created automatically. For each *Zone* it is necessary to define a proper type of a boundary condition. From *Zone* list select the *wall* name and make sure that the type of boundary condition is set up as *Wall* (Fig. E1-9b).

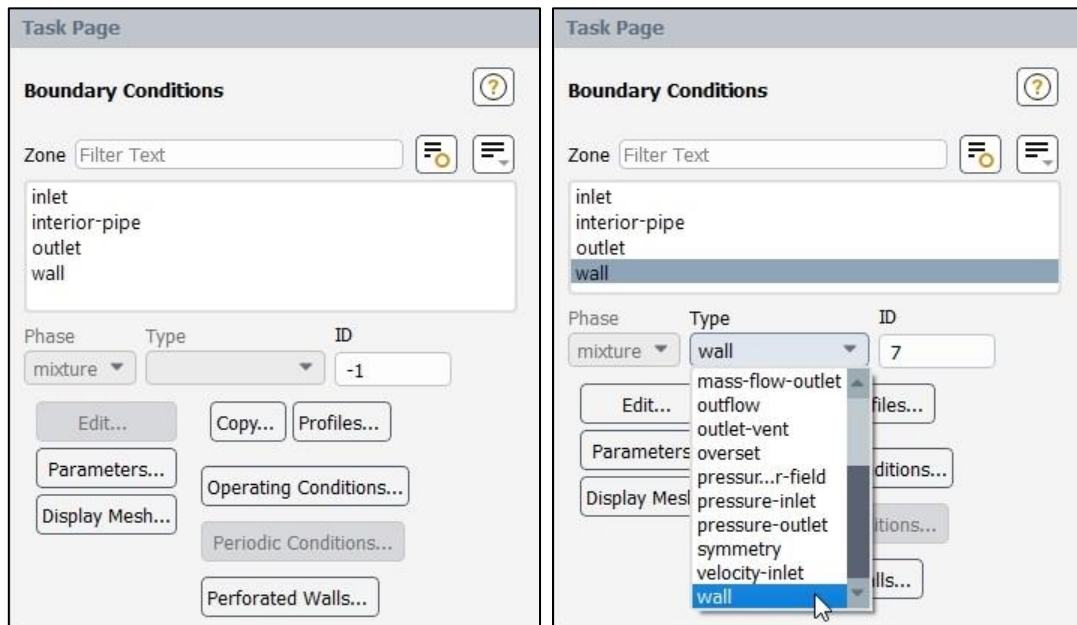


Figure E1-9. *Boundary Condition Task Page* and definition of the boundary condition type.

With the same procedure set up the remaining boundary conditions as:

1. for the *interior-pipe* zone → the *interior* boundary condition,
2. for the *outlet* zone → the *pressure-outlet* boundary condition,

- for the *inlet* zone → the *velocity-inlet* boundary condition. We also need to enter the velocity of the fluid at the pipe entrance. In this purpose click *Edit...* and in the *Velocity Inlet* dialog box (Fig. E1-10) enter the *Velocity Magnitude [m/s]* as $w = 0.001005$ [m/s]. This value was calculated from the Reynolds number definition for the specified in the example values of the Reynolds number, pipe diameter and water properties (density and viscosity, Fig. E1-7).

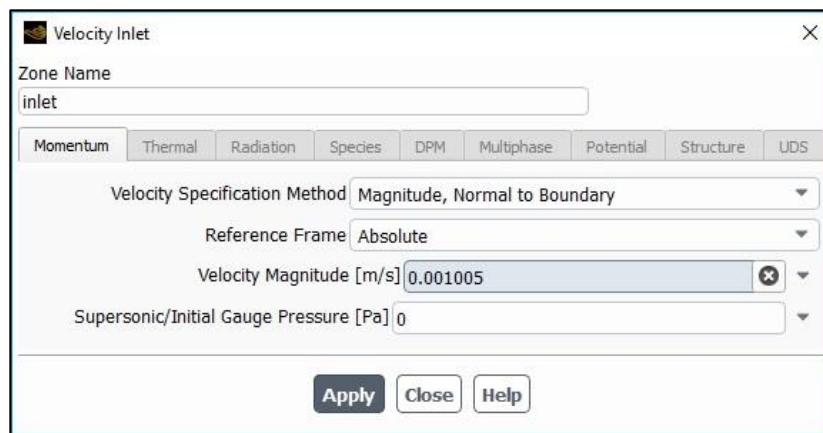


Figure E1-10. *Velocity Inlet* dialog box.

F. *Solution – Methods*

Open the *Solution Methods Task Page* (Fig. E1-11) by double-click the *Methods* in the *Outline View Tree*. 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*,
- Other options are disabled.

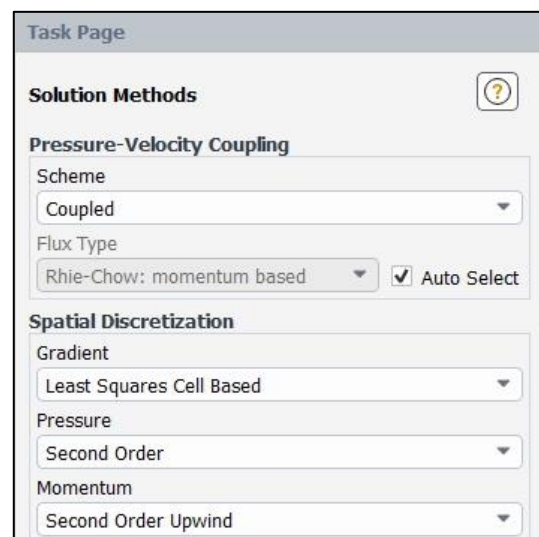


Figure E1-11. *Solution Methods* Task Page.

G. *Solution – Monitors*

Expand the *Monitors* tab in the *Outline View Tree* by clicking “plus” icon on the left side of this tab. Then open the *Residual Monitors* dialog box (Fig. E1-12) by double-click the *Residual* in the *Outline View Tree*. Make sure that all equations are monitored and their convergence is checked, as it is shown in the Fig. E1-12. Set all *Absolute Criteria* as $1e-13$ and close the window by clicking OK.

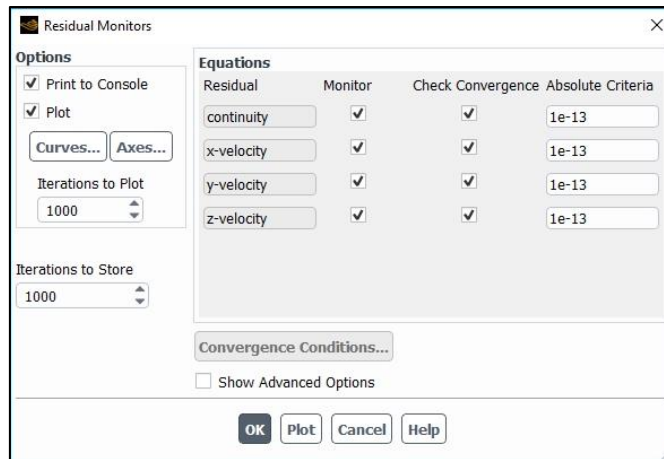


Figure E1-12. Residual Monitors dialog box.

H. Solution – Initialization

Open the *Solution Initialization Task Page* (Fig. E1-13a) by double-click the *Initialization* in the *Outline View Tree*. Here, the user can choose between two types of initialization:

1. *Hybrid Initialization* – provides a quick approximation of the flow field by solving Laplace’s equation to estimate a flow and pressure fields.
2. *Standard Initialization* – sets all mesh cells to a single starting value. The user should enter values of parameters that are close to the final solutions in order to help the solution converge faster. The options *Compute from inlet* takes inlet conditions that was defined in the boundary conditions tab and applies them to the whole mesh as a starting guess.

In this example we use the *Standard Initialization* with enabled *Compute from inlet* option (Fig. E1-13b). After selecting the initialization technique, click *Initialize* button.

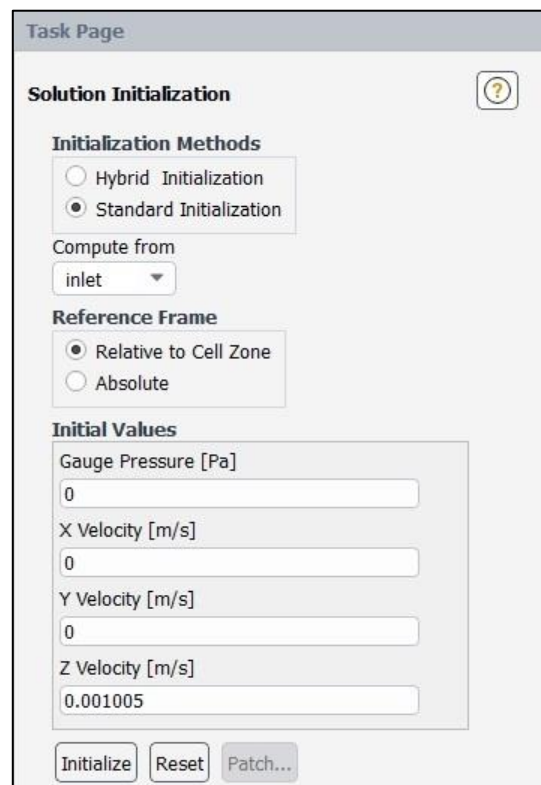
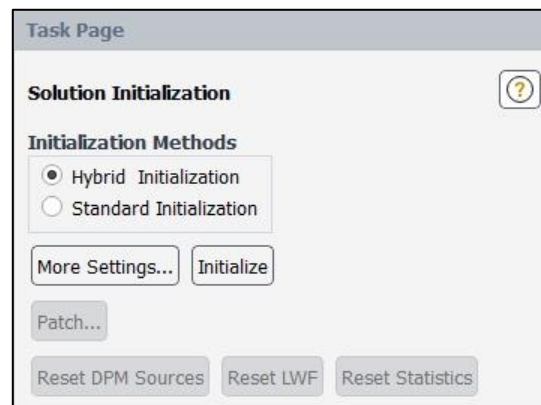


Figure E1-13. Solution Initialization Task Page.

I. Solution – Run Calculation

Open the *Run Calculation Task Page* (Fig. E1-14) by double-click the *Run Calculation* in the *Outline View Tree*. Before run calculation press the *Check Case...* button to make sure everything was set up correctly. You should get the information “*No recommendation to make at this time*”, that means the setup is good.

Set the *Number of Iterations* as 200 and click *Calculate*. During the calculations in the main window you will see the *Scaled Residuals* for each of the equation parameters (in this case *continuity*, *x-velocity*, *y-velocity*, *z-velocity*) as in Fig. E1-15. The solution will be stopped by *Ansys Fluent* when all residuals reach their specified values (here $1e-13$) or after 200 iterations. After complete the iteration you will see the dialog box with the information “*Calculation complete*”.

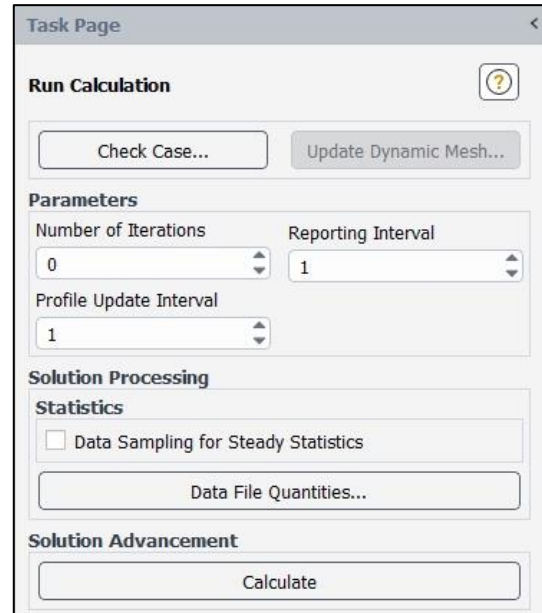


Figure E1-14. Run Calculation Task Page.

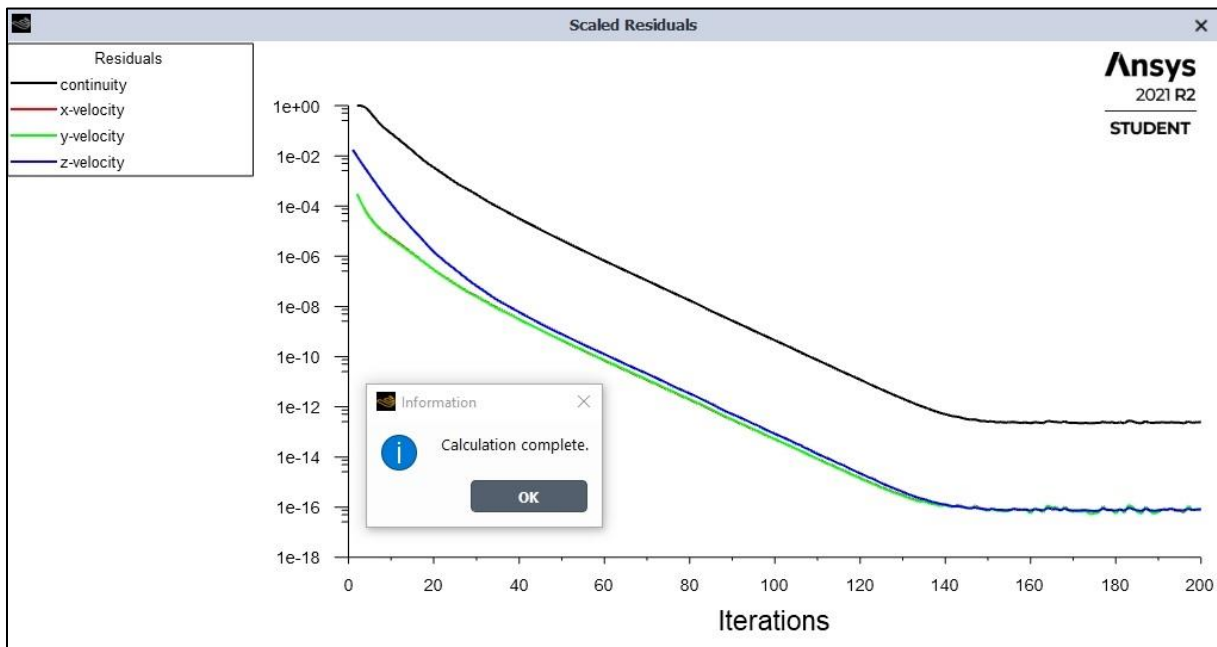


Figure E1-15. Residuals for the converged solution.

Since the solution in the above example is convergent we can start displaying the results. It can be done in both *Ansys Fluent* and *CFD Post* programs. Here we will use the *Ansys Fluent*.

1.4. Results

In this example as a results we will show contours and profiles of fluid velocity. In order to create contours of velocity double-click the *Contours* in the *Outline View Tree*. A *Contours* dialog box will open (Fig. E1-16). The contours of velocity will be presented at three locations: at the inlet, outlet and in the mid-plane of the pipe. Two of them (inlet and outlet) are visible in the *Contours* dialog box *Surfaces* list. The mid-plane has to be created.

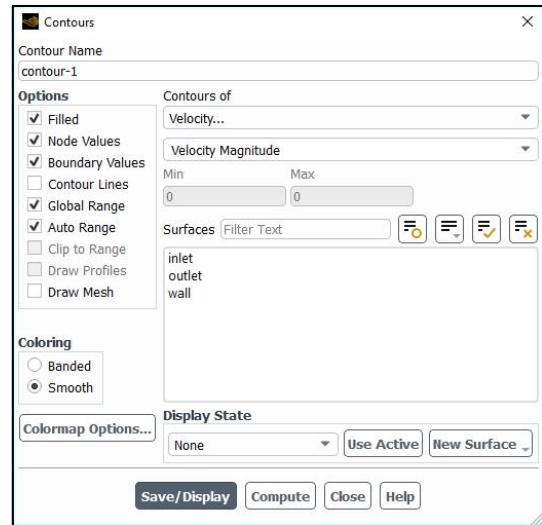


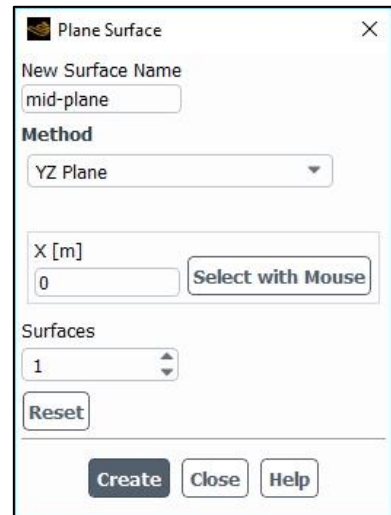
Figure E1-16. *Contours* dialog box.

In this purpose from the *Contours* dialog box choose *New Surface* → *Plane...* A *Plane Surface* dialog box will open. Here enter the following settings (Fig. E1-17):

1. *New Surface Name* → *mid-plane*,
2. *Method* → *YZ Plane*,
3. *X* → *0*,
4. *Surfaces* → *1*,

and confirm by clicking *Create*. Note that during creation of the new plane its shaded view is visible in the main window.

Figure E1-17. *Contours* dialog box.



After complete this operation a new plane will appear on the *Surfaces* list. Come back to the *Contours* dialog box and enter the following settings:

1. *Contours of* → *Velocity...* → *Velocity Magnitude*,
2. *Surfaces* → *mid-plane*,
3. *Coloring* → *Smooth*,
4. Options such as *Filled*, *Node Values*, *Boundary Values*, *Global Range* and *Auto Range* should be enabled.

Click *Save/Display* button. A created contours of magnitude velocity in a mid-plane will appear in the main window. You can set up the proper view using the *Pointer* toolbar (rotate, pan, zoom options) or by clicking directly on the axes visible in the right bottom corner of main

window. In a similar way create contours of magnitude velocity at the inlet and outlet surfaces. All these results are presented in Fig. E1-18.

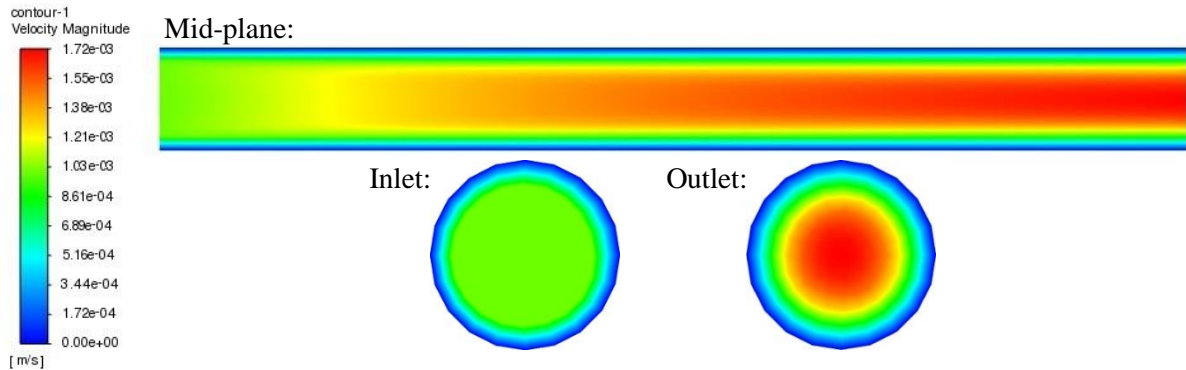


Figure E1-18. Contours of fluid velocity magnitude in the mid-plane, at the inflow and outflow surfaces.

In order to create profiles of velocity double-click the *XY Plot* in the *Outline View Tree*. A *Solution XY Plot* dialog box will open. The profiles of velocity will be presented along the diameter of the pipe at three locations starting from the inlet: 0.5 m, 2.5 m and 4.5 m. They have to be created by choosing *New Surface* → *Line/Rake...* A *Line/Rake Surface* dialog box will open. Here enter the following settings (Fig. E1-19a):

1. *New Surface Name* → *line-0.5*,
2. *Type* → *Rake*,
3. *Number of Points* → 25,
4. *End Points* → $x0 [m] = 0, \quad y0 [m] = -0.25, \quad z0 [m] = 0.5,$
 $x1 [m] = 0, \quad y1 [m] = 0.25, \quad z1 [m] = 0.5,$

and confirm by clicking *Create*. In analogous way create two remaining lines with the settings showed in Figs. E1-19b, c.

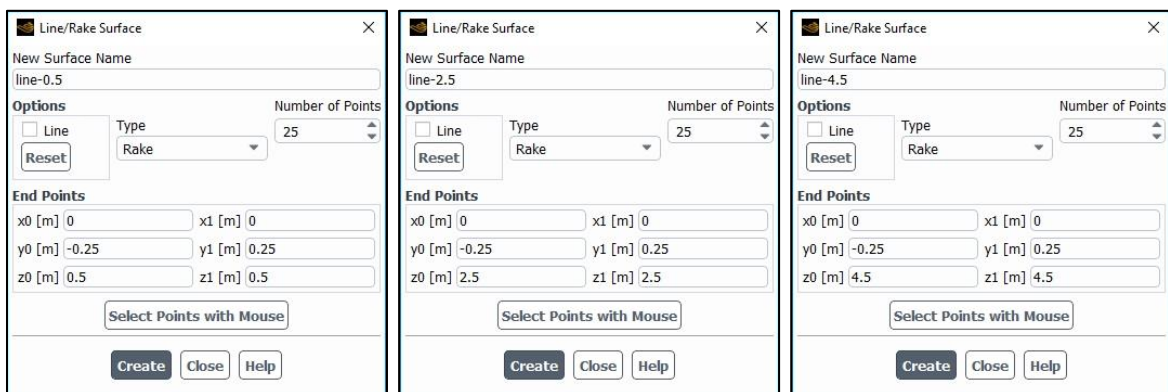


Figure E1-19. Settings of the three line locations needed to *XY Plot* creation.

After complete this operation new lines will appear on the *Surfaces* list. Come back to the *Solution XY Plot* dialog box and enter the following settings (Fig. E1-20):

1. *Plot Direction* → $X = 0, Y = 1, Z = 0,$

2. Y Axis Function → Velocity... → Velocity Magnitude,
3. X Axis Function → Direction Vector,
4. Surfaces → line-0.5, line-2.5, line-4.5 (in order to select multiple objects press and hold Ctrl while you click on them),
5. Options such as Node Values, Position on X Axis should be enabled.

Confirm by clicking *Save/Plot* and close the window. As a result, in the main window you will see the graph with profiles of velocity magnitude along the diameter in three subsequent locations (Fig. E1-21).

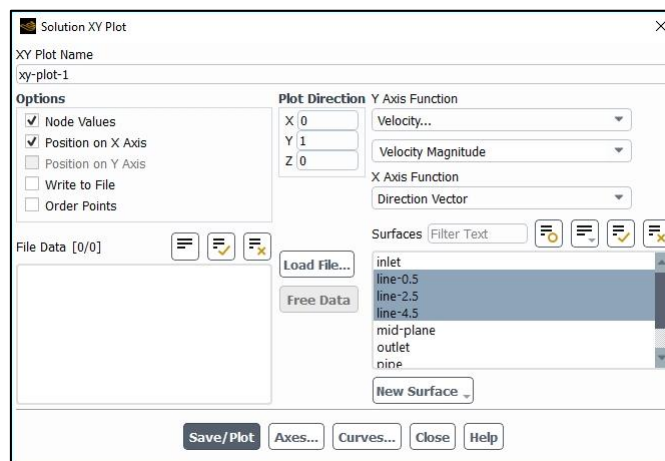


Figure E1-20. Settings in the *Solution XY Plot* dialog box.

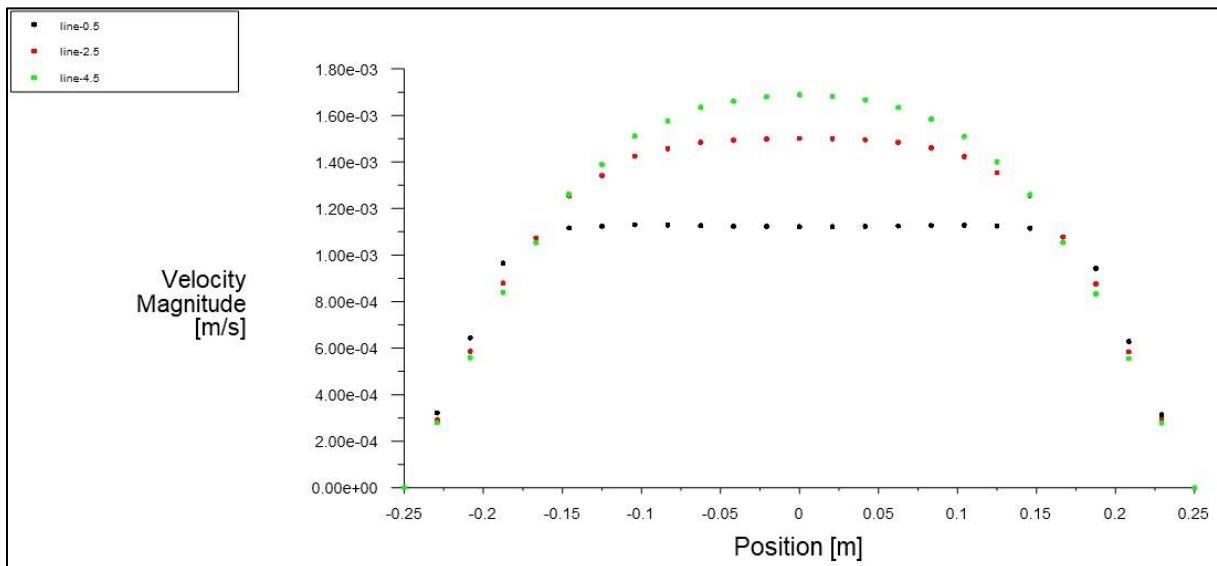


Figure E1-21. Profiles of velocity magnitude along the pipe diameter in three locations.

Do not close Fluent software. We will start second exercise from this point.

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.

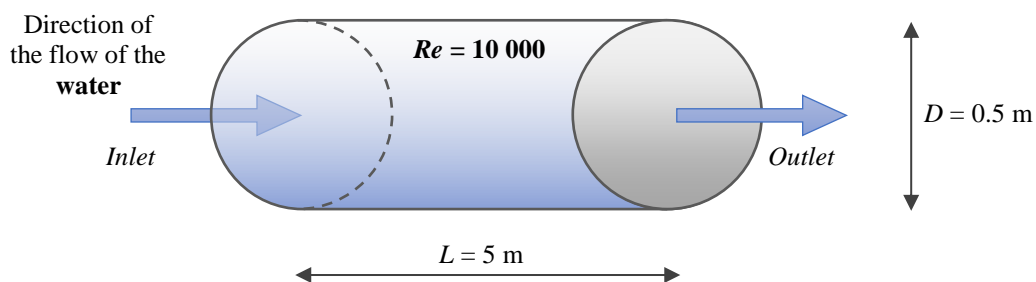


Figure E2-1. Problem description.

2.3. Geometry and Mesh

Geometry and mesh representing the flow domain in this example are the same like in the previous example. Only value of the Reynolds number has changed.

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. All the new settings will be described with details.

A. 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-ε 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. E2-2). 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.

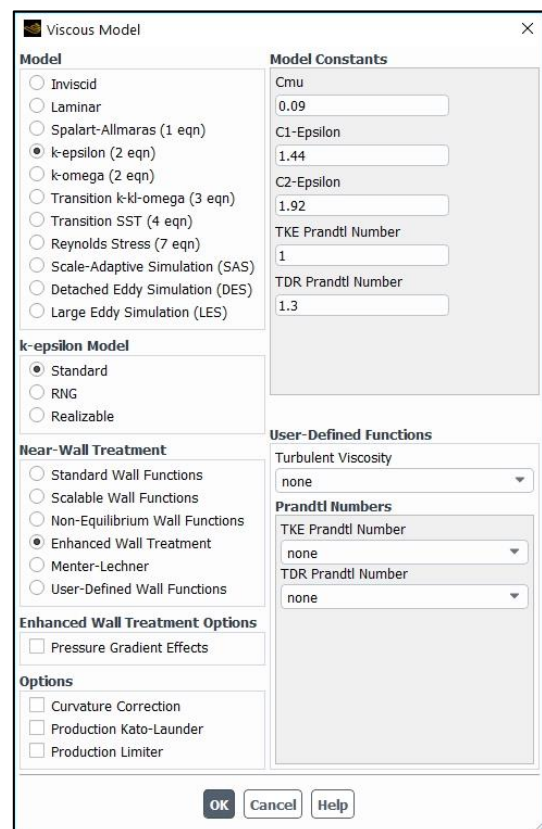


Figure E2-2. *Viscous Model* dialog box with enabled *k-epsilon* model.

B. 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].

C. 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* → *Coupled*, *Flux Type* → *Auto Select*, *Gradient* → *Least Squares Cell Based*, *Pressure* → *Second Order*, *Momentum* → *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, ϵ** , are calculated. For both these parameters select from the dropdown list the *Second Order Upwind* scheme of spatial discretization.

D. 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 k and ϵ – parameters that come from the turbulence model.

E. Solution – Initialization

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

F. 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. E2-3). 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 six equation parameters (Fig. E2-4).

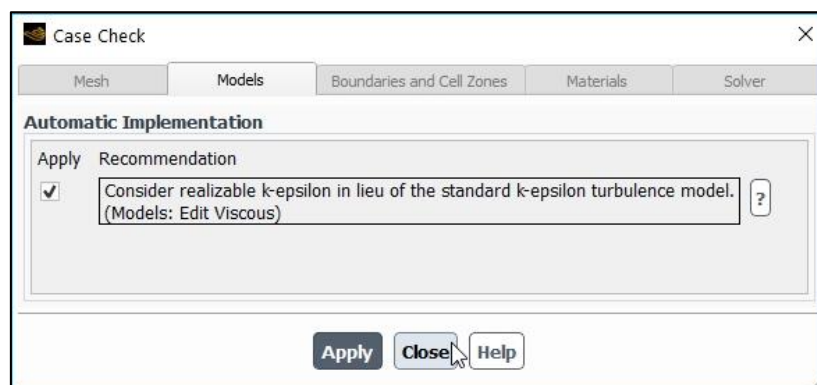


Figure E2-2. Case Check dialog box with recommendation.

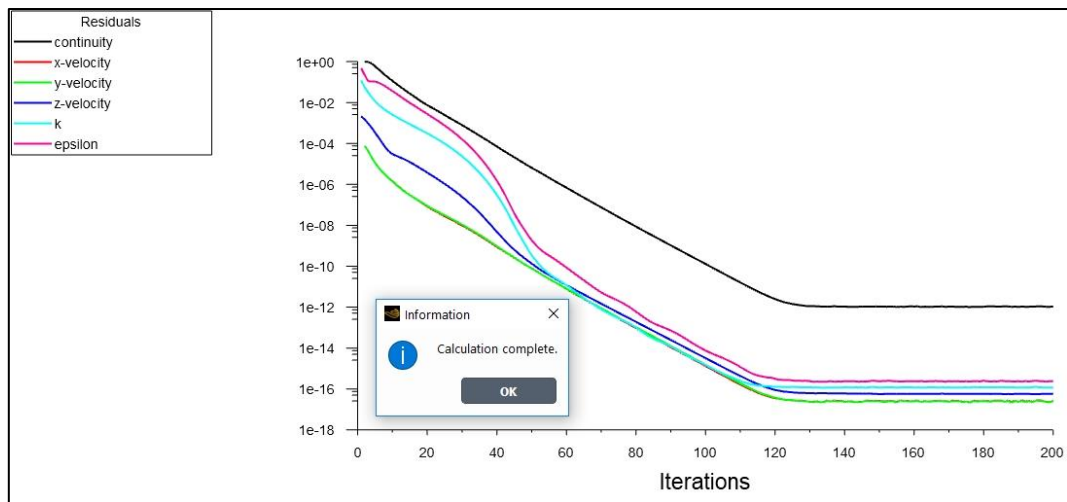


Figure E2-4. Residuals for the converged solution.

2.5. Results

In this example results will be presented as a contours and profiles of the fluid velocity as well as contours of the turbulent kinetic energy and turbulent dissipation rate.

In order to present velocity contours (Fig. E2-5) and profiles (Fig. E2-6) use mid-plane and three lines generated previously. 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. E2-7). Next, from the dropdown list first the *Turbulent Kinetic Energy (k)*, and then the *Turbulent Dissipation Rate (epsilon)* were selected. Results are presented in Figure E2-8.

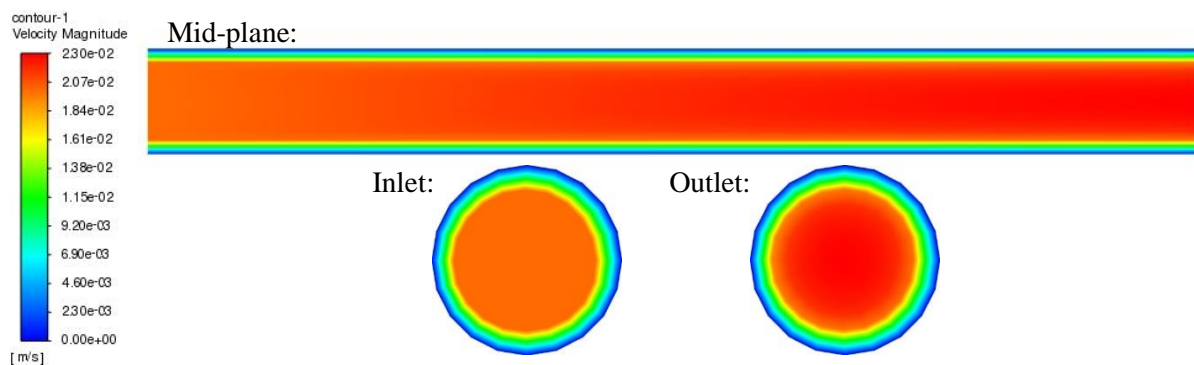


Figure E2-5. Contours of fluid velocity magnitude in the mid-plane, at the inflow and outflow surfaces.

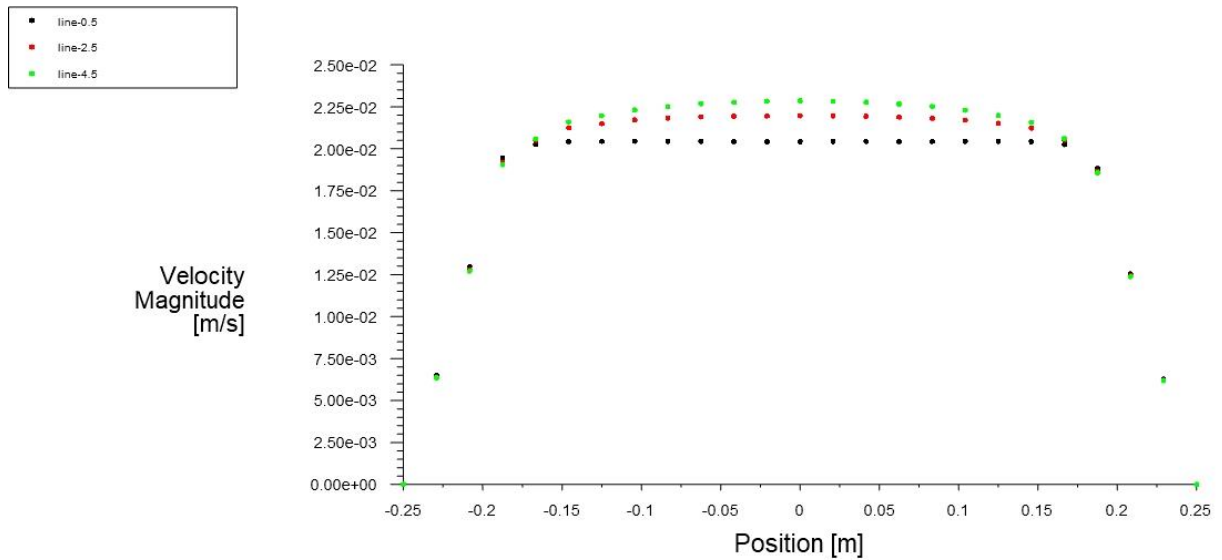


Figure E2-6. Profiles of velocity magnitude along the pipe diameter in three locations.

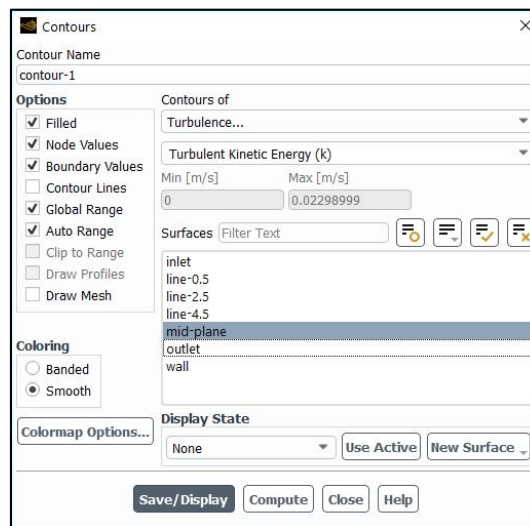


Figure E2-7. *Contours* dialog box set up for the contours of the turbulent kinetic energy.

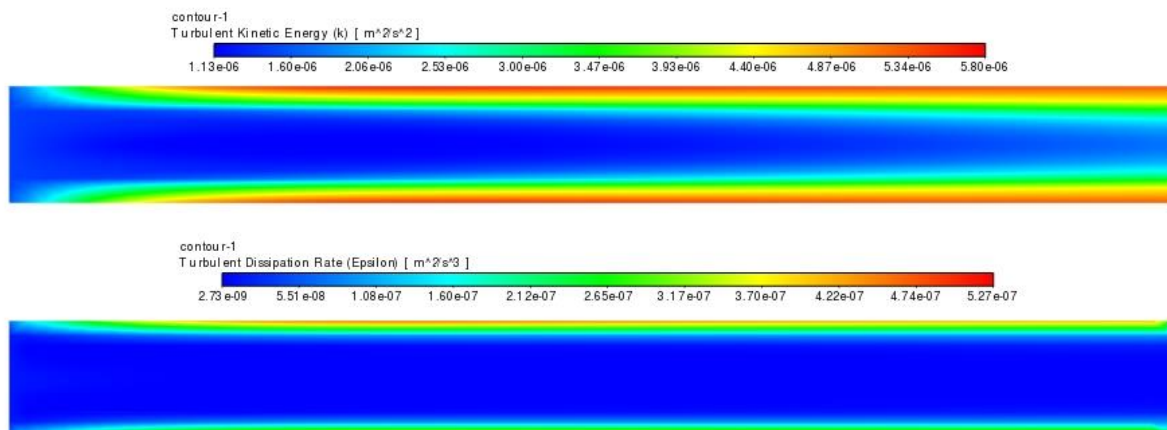


Figure E2-8. Contours of turbulent kinetic energy and turbulent dissipation rate in the pipe mid-plane.