## 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.

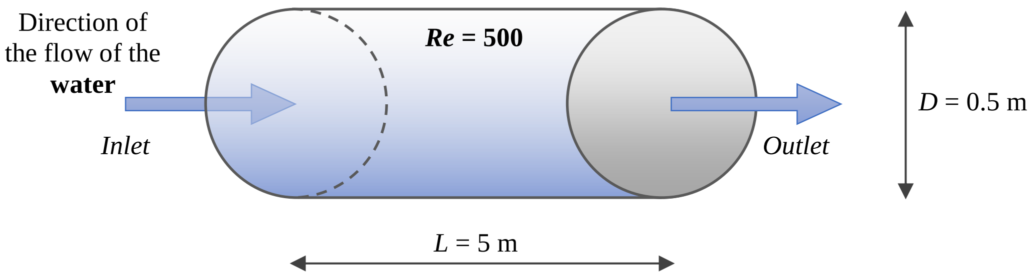


**Figure E1-1.** Problem description.

### 1.3. Geometry

Into the *Project Schematic* window insert a *Fluid Flow (Fluent)* module from *Analysis Systems* tab. Open the *Ansys DesignModeler* as described in the *Introduction*. There are different methods of the geometry drawing. In this example we will use the *Primitives* method. From the main menu choose *Create* → *Primitives* → *Cylinder* (Fig. E1-2a). A new figure (*Cylinder1*) appeared in the *Tree Outline*, and its details such as basic plane, origin or dimensions are visible in the *Details View* window placed below the *Tree Outline* (Fig. E1-2b). Also a grey wireframe of the figure is visible in the *Graphics* window (Fig. E1-2b). In the

*Details View* window enter the following parameter values: *Axis Z Component* – 5 m and *Radius* – 0.25 m (Fig. E1-2c). Important information – *Ansys DesignModeler* uses comas instead of dots. After entering these values, click the *Generate* button from the top menu (Fig. E1-2d). If everything has been done correctly, a shaded figure will appear in the *Graphics window* and a corresponding green check mark will appear next to the cylinder name in *Tree Outline* (Fig. E1-2e). At the end save the project by selecting from the main menu *File* → *Save Project*. You can close *Ansys DesignModeler*.

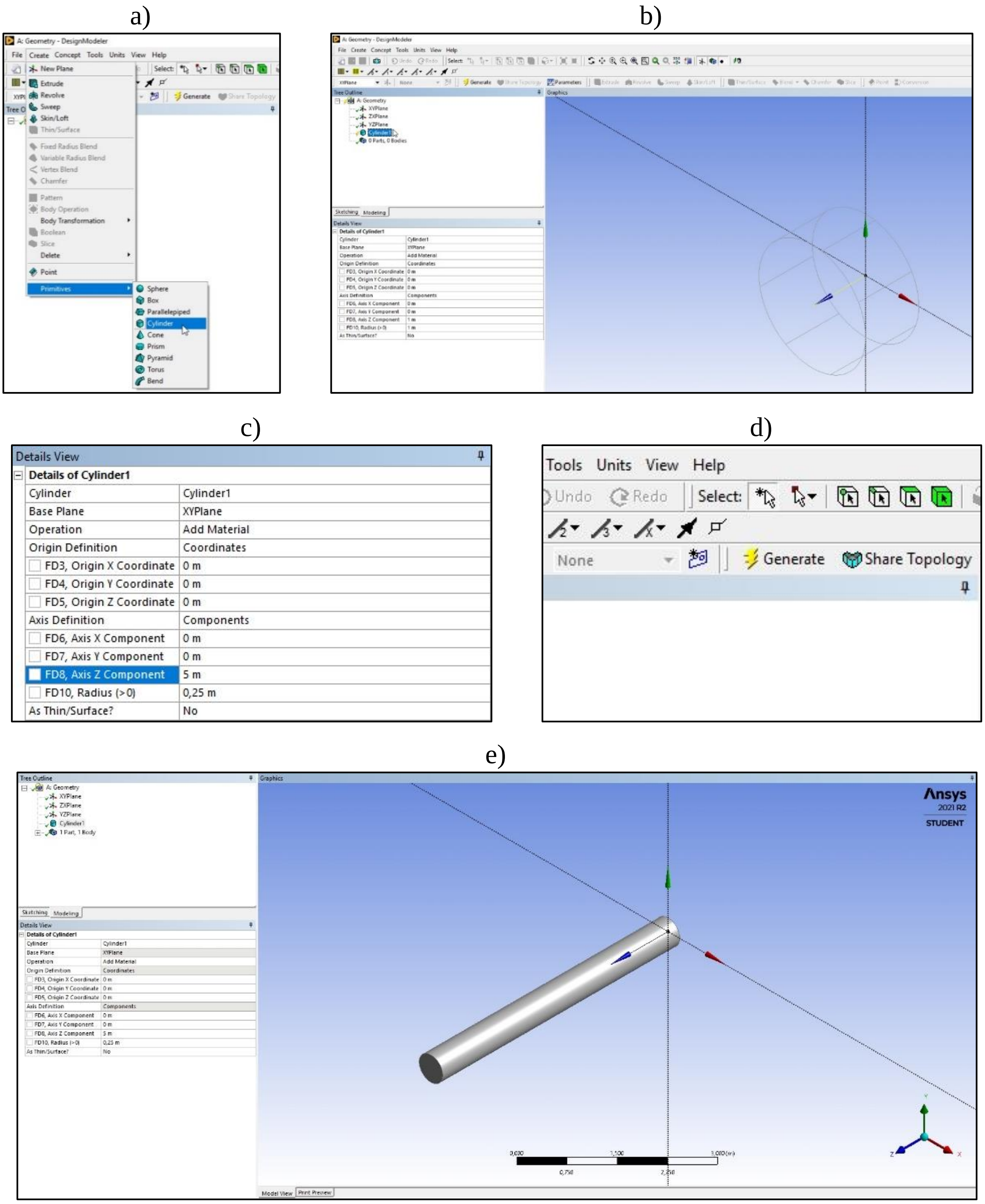


**Figure E1-2.** Steps of the geometry drawing.

## 1.4. Mesh

Open the *Ansys Meshing* as described in the *Introduction*. After opening you will see a geometry window with the shape of the geometry drawn in the *Geometry* module in the previous step (Fig. E1-3). Over this window there is a main menu as well as different toolbars used, among others, to establish and edit the mesh settings, set up the *Named Selection* or set up the view of the flow domain. On the left side there are also the *Tree Outline* and *Details View*. *Tree Outline* is the object tree that represents outline view of the whole project and provides access to object's context menus. *Details View*, similar like in *Ansys DesignModeler*, contains details about each object in the *Tree Outline*.

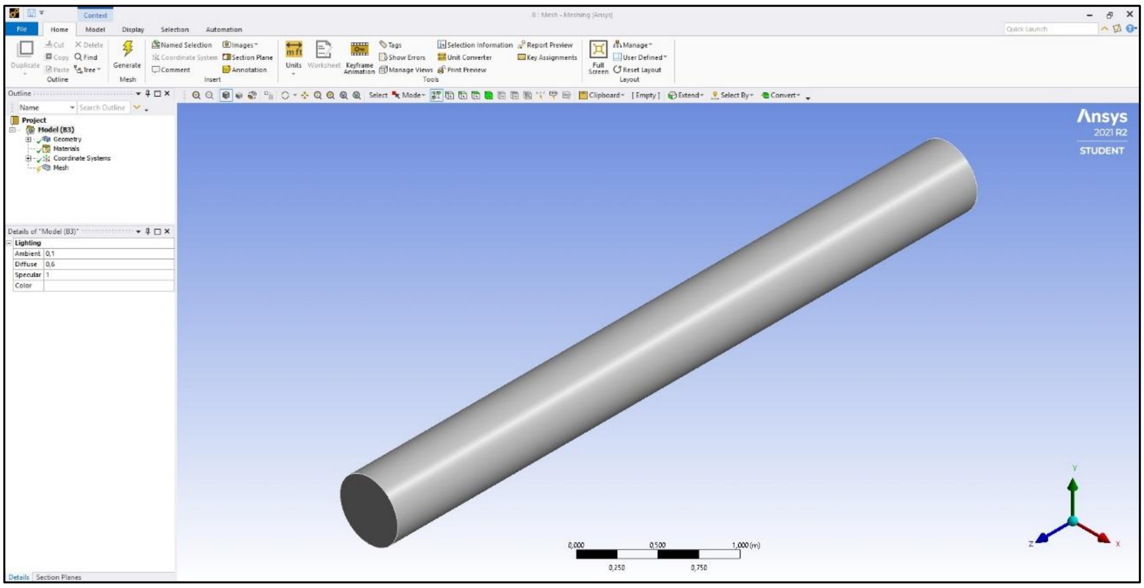


**Figure E1-3.** Ansys Meshing basic view.

Before the mesh generation we will change the name of the geometry and specify the material. In this purpose in the *Tree Outline* expand the *Geometry*, select the *Solid* with the RMB and choose *Rename* option (Fig. E1-4a). Enter the "*Pipe*" name. Now, in the *Details of "Pipe"* select the *Material* as *Fluid* (Fig. E1-4b). Please note that both above settings can be done in *Ansys Meshing* or *Ansys DesignModeler* module.

Now choose the *Mesh* from the *Tree Outline*. The shape of the flow domain in the present example is quite simple, therefore it is possible to generate a structured mesh. In the *Details of "Mesh"* select the *Defaults* → *Physics Preference* as *CFD* and *Solver Preference* as *Fluent* (Fig. E1-4c). The same as in the *Ansys DesignModeler* each operation has to be ended with generation, therefore press the *Generate Mesh* button that is visible in the top *Home Menu*. The mesh will generate automatically. In the *Details of "Mesh"* you can check its *Quality* and *Statistics* (Fig. E1-4d). Recall form the chapter *5.1.2. Mesh Quality* that in the case of the hexahedral element the mesh is considered good for skewness value less than 0.8. Here, the maximum value of skewness does not exceed 0.38. However, the number of elements is quite low – 3968 elements. We will increase this number by expanding in the *Details of "Mesh"* the *Sizing* tab and changing the default value of *Growth Rate* from 1.2 for 1.02 (Fig. E1-4e). After regenerate the mesh, the number of element in the grid increases to 5888 and the maximum value of skewness does not exceed 0.36 (Fig. E1-4f).

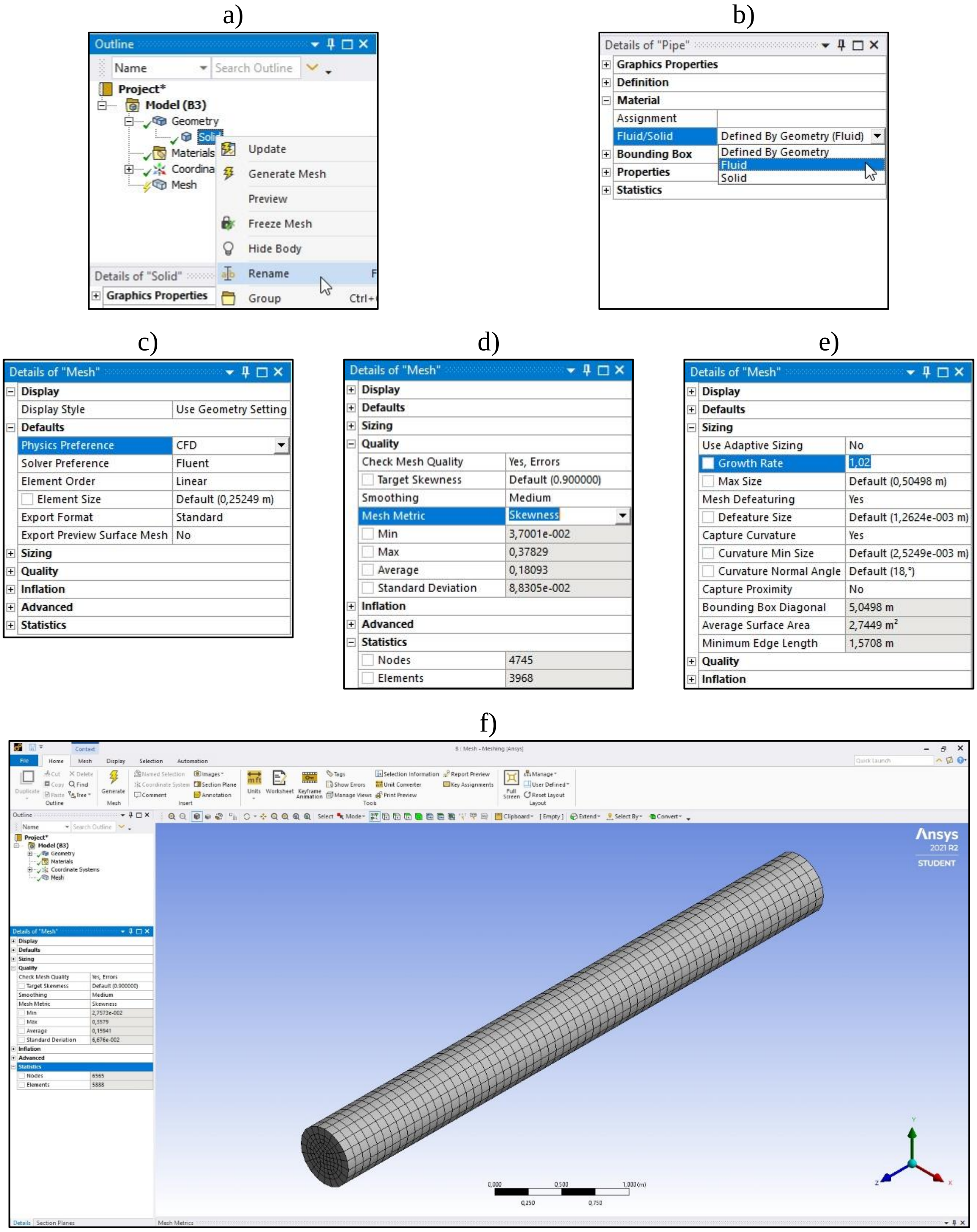


**Figure E1-4.** Steps of the mesh generation.

The last step in the *Ansys Meshing* software is creation of *Named Selection*. In the analysed flow task, we will create three *Named Selections* on the surfaces – one for the inlet, one for the outlet and one for the wall. First, choose *Face* option from the *Graphics Toolbar* (Fig. E1-5a) and then select a proper surface from the geometry by clicking on them in the *Geometry Window*. The chosen surface is highlighted by green colour (Fig. E1-5b). Click again on this surface with RMB and choose *Create Named Selection…* In the new window that will appear enter a name for the chosen surface, here it is *Outlet* (Fig. E1-5c). Repeat this step for two remaining surfaces – *Inlet* and *Wall*. If you need to move or turn your geometry, you can do this using options collected in the *Graphics Toolbar*, such as *Rotate*, *Pan*, *Zoom*, *Box Zoom*, *Zoom To Fit*. All created *Named Selections* are visible in the *Tree Outline*. An active *Named Selection* is highlighted with red colour in the *Geometry Window* (Fig. E1-5d) and its characteristics are collected in the *Details* window. Save the project and close *Ansys Meshing*.

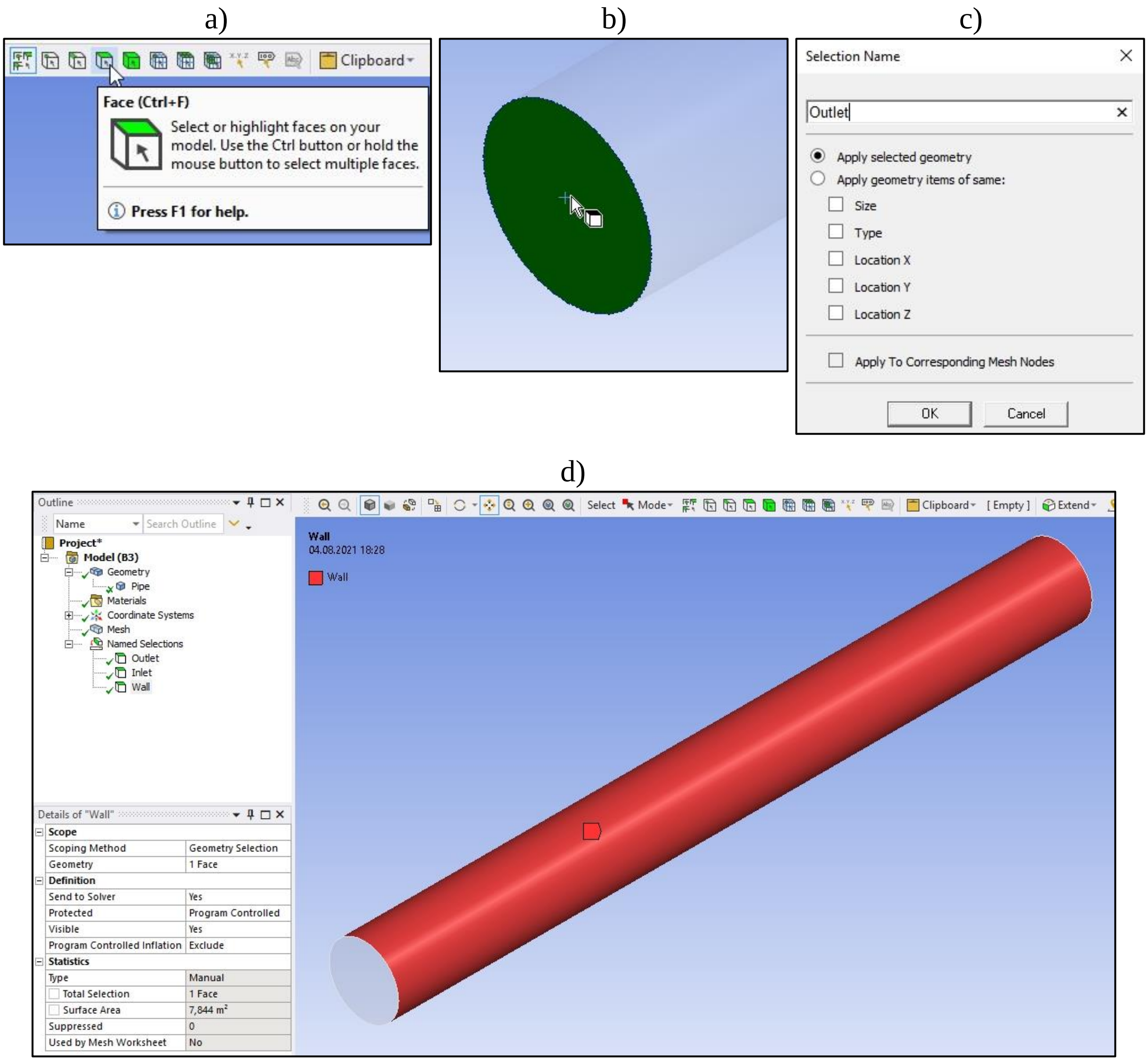


**Figure E1-5.** Creation of the *Named Selections*.

## 1.5. Simulations

Open the *Ansys Fluent* software by a double-click the *Setup* in the *Fluid Flow (Fluent)* module. During the first start of *Ansys Fluent*, the *Fluent Launcher* (Fig. E1-6) 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*.

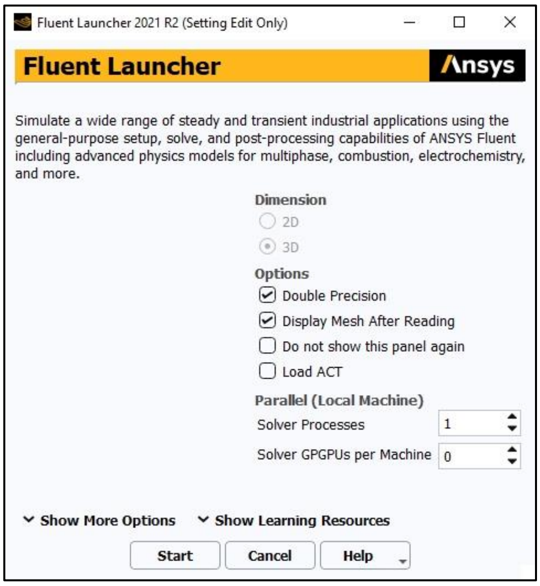


**Figure E1-6.** Fluent Launcher.

The *Ansys Fluent* window (Fig. E1-7) 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.

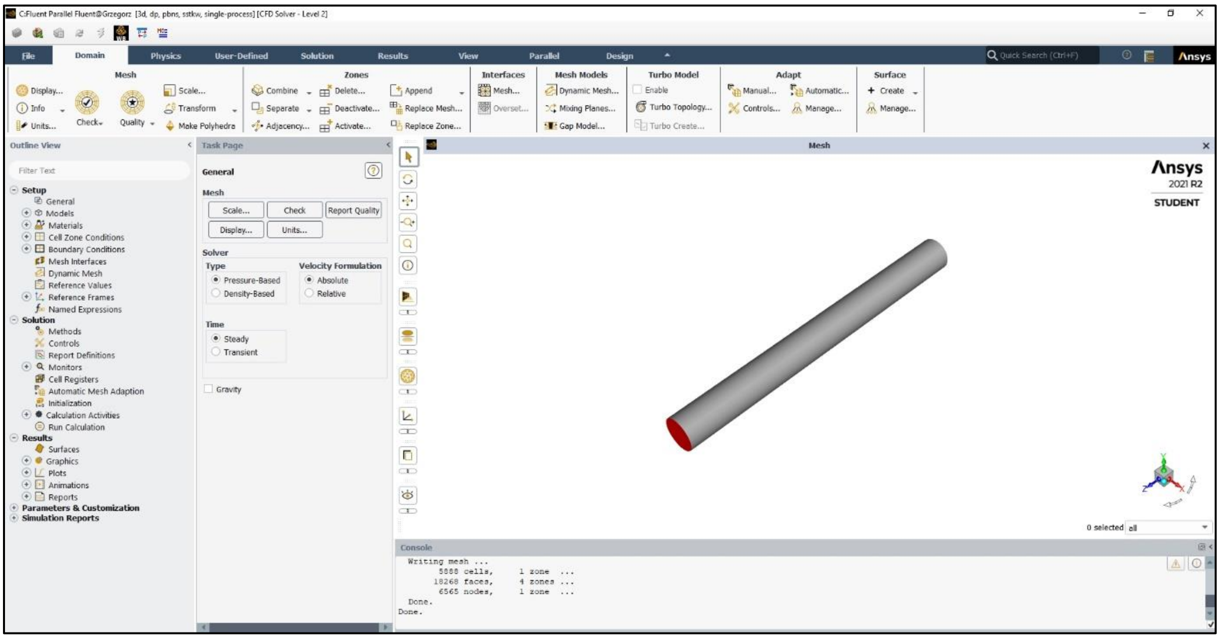


**Figure E1-7.** *Ansys Fluent* layout.

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. E1-8) 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.

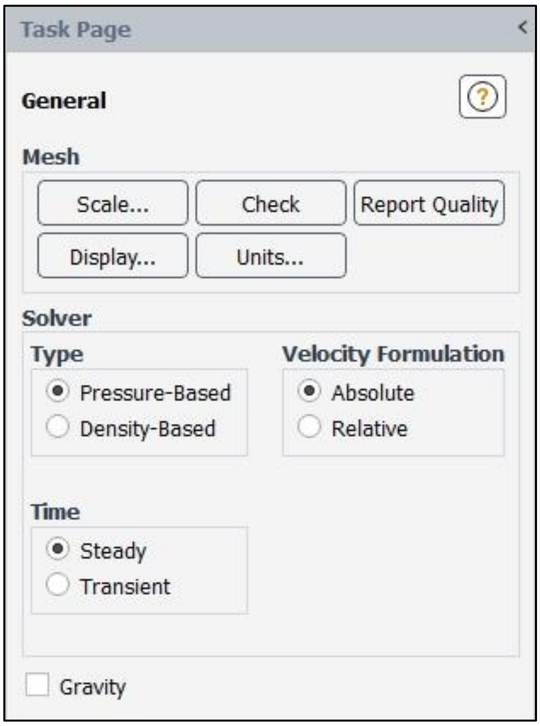


**Figure E1-8.** *General* Task Page.

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. E1-9) by double-click the *Models* in the *Outline View Tree*. Make sure that the *Laminar* model is selected within the *Viscous* tab.

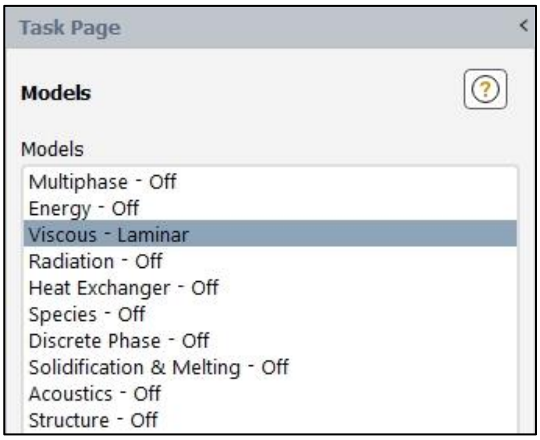


**Figure E1-9.** *Models* Task Page.

C. ***Setup*** – ***Materials***

Open the *Materials Task Page* (Fig. E1-10) 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. E1-11a). 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. E1-11b) 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.

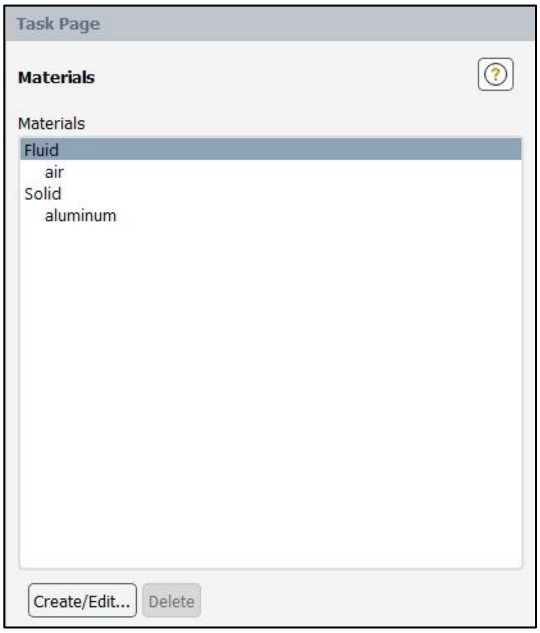


**Figure E1-10.** *Materials* Task Page.

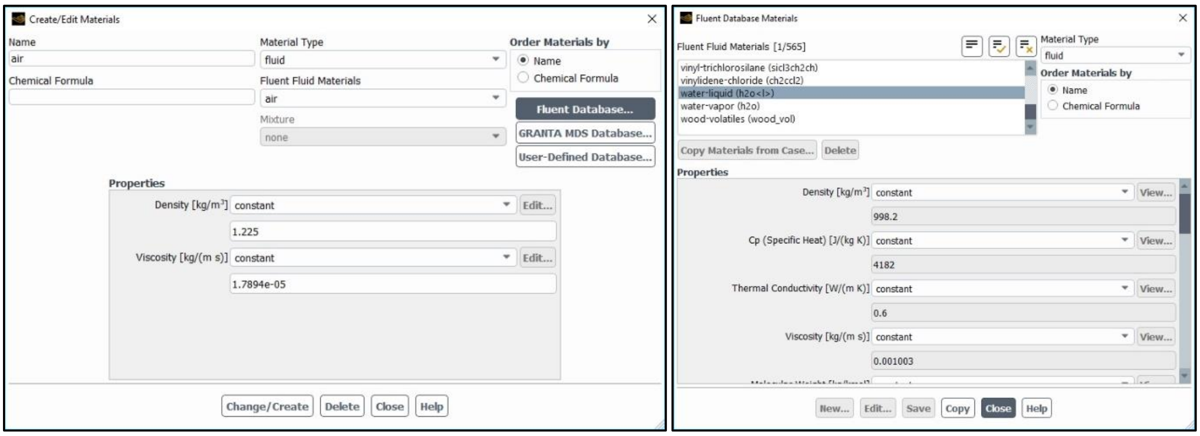


**Figure E1-11.** Adding a new material from the *Fluent Database*.

D. ***Setup*** – ***Cell Zone Conditions***

Open the *Cell Zone Conditions Task Page* (Fig. E1-12a) 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. E1-12b) 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.

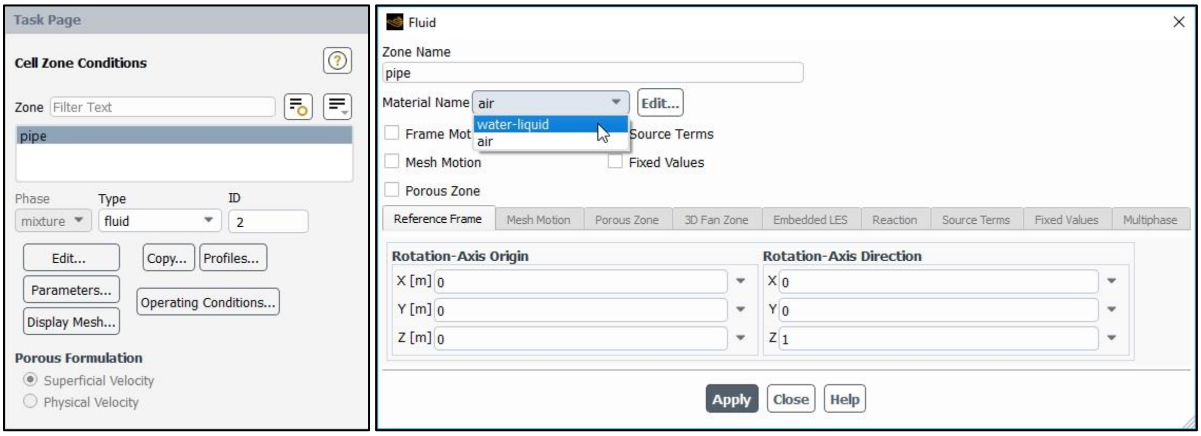


**Figure E1-12.** Setup of the *Cell Zone Conditions.*

E. ***Setup – Boundary Conditions***

Open the *Boundary Conditions Task Page* (Fig. E1-13a) 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. E1-13b).

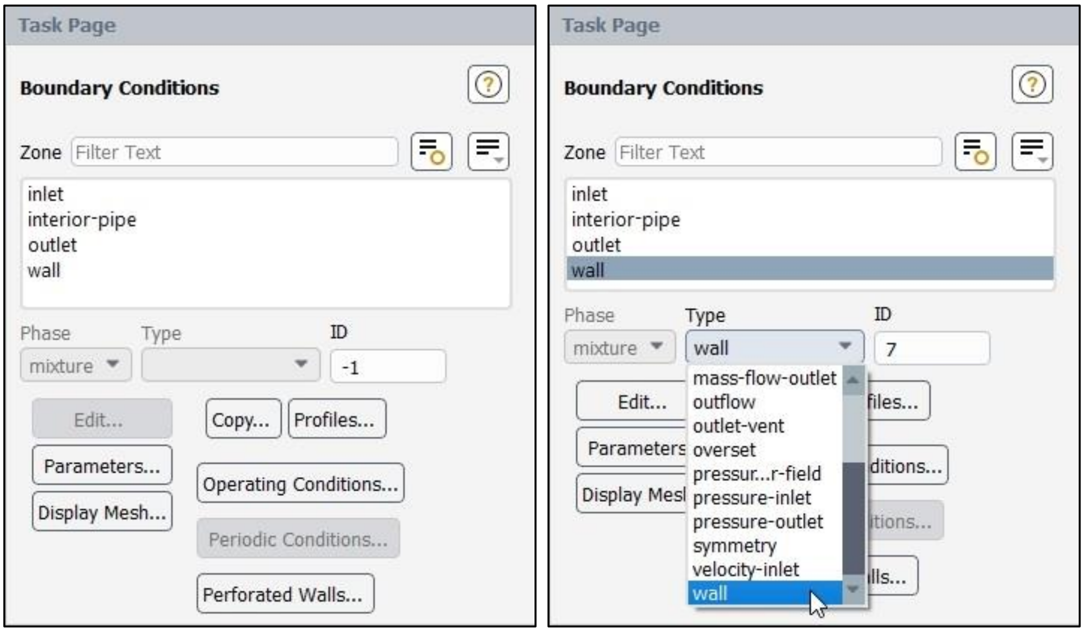


**Figure E1-13.** *Boundary Condition* Task Page and definition of the boundary condition type.

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,

3. 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. E1-14) 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. E1-11).

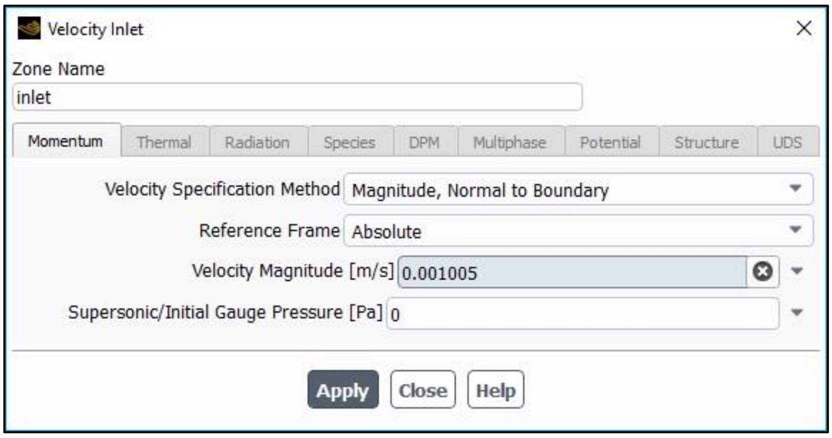


**Figure E1-14.** *Velocity Inlet* dialog box.

F. ***Solution*** – ***Methods***

Open the *Solution Methods Task Page* (Fig. E1-15) by double-click the *Methods* in the *Outline View Tree*. Ensure that the options within this page are set up as:

1. ***Scheme*** → ***Coupled***,
2. ***Flux Type*** → ***Auto Select***,
3. ***Gradient*** → ***Least Squares Cell Based***,
4. ***Pressure*** → ***Second Order***,
5. ***Momentum*** → ***Second Order Upwind***,
6. Other options are disabled.

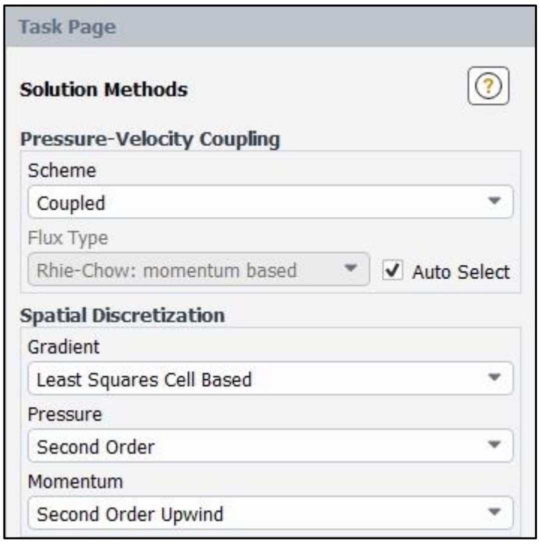


**Figure E1-15.** *Solution Methods* Task Page.

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. E1-16) 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. E1-16. Set all *Absolute Criteria* as 1e-13 and close the window by clicking OK.

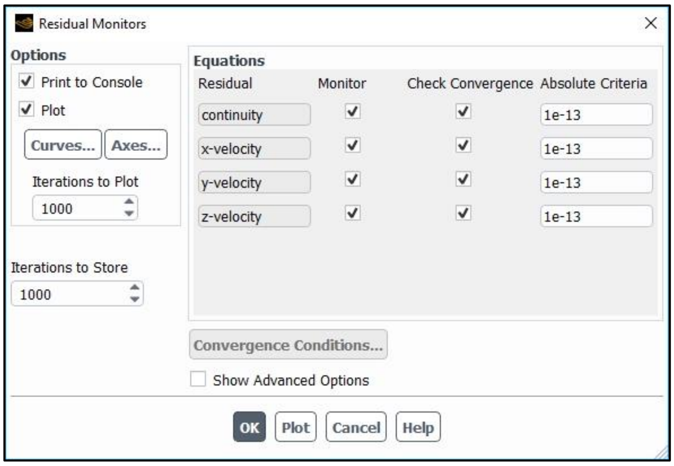


**Figure E1-16.** *Residual Monitors* dialog box.

H. ***Solution*** – ***Initialization***

Open the *Solution Initialization Task Page* (Fig. E1-17a) 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. E1-17b). After selecting the initialization technique, click *Initialize* button.

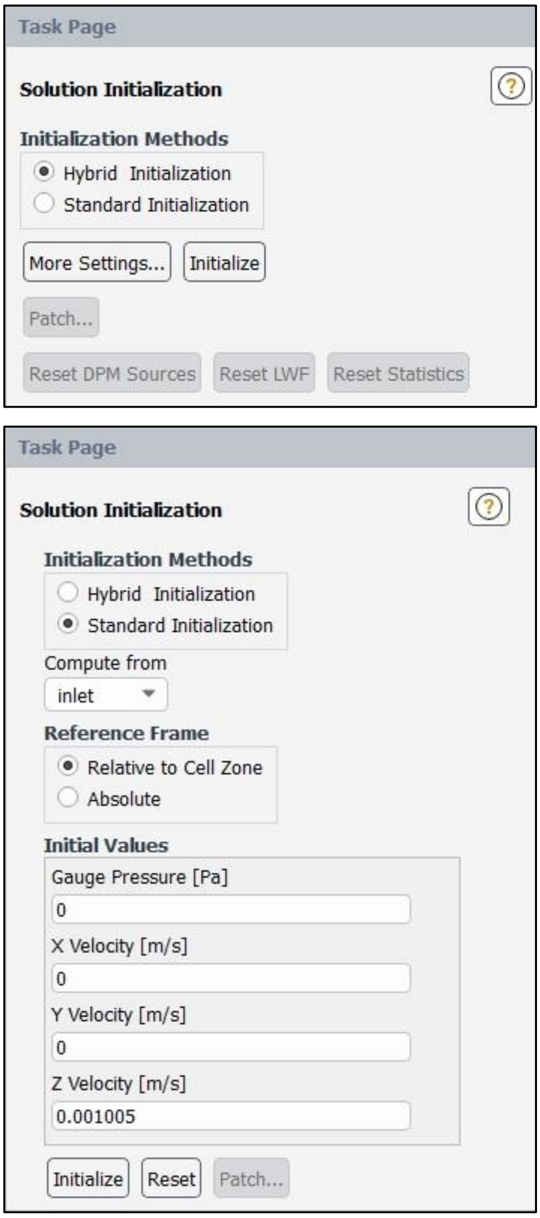


**Figure E1-17.** *Solution Initialization* Task Page.

I. ***Solution*** – ***Run Calculation***

Open the *Run Calculation Task Page* (Fig. E1-18) 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. E1-19. 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*".

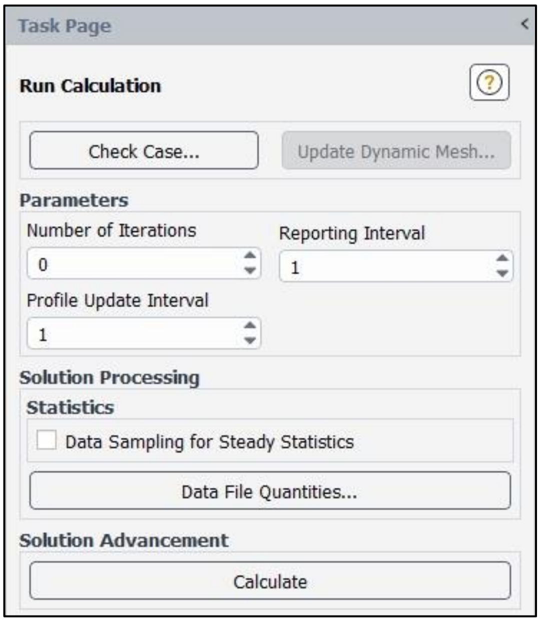


**Figure E1-18.** *Run Calculation* Task Page.

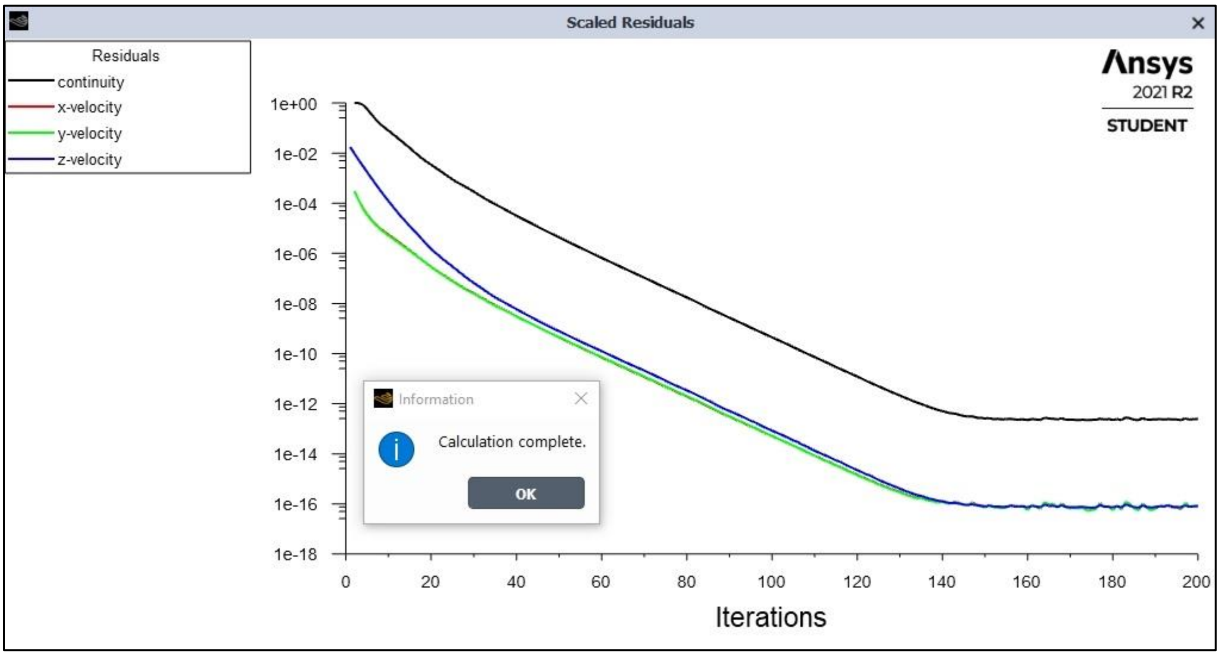


**Figure E1-19.** Residuals for the converged solution.

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*.

## 1.6. 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. E1-20). 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.

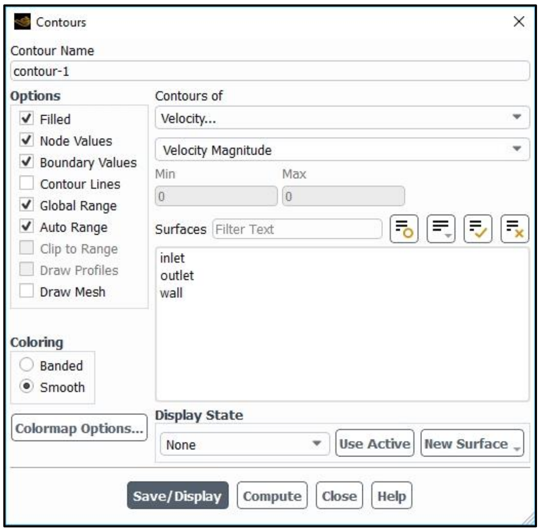


**Figure E1-20.** *Contours* dialog box.

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. E1-21):

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.

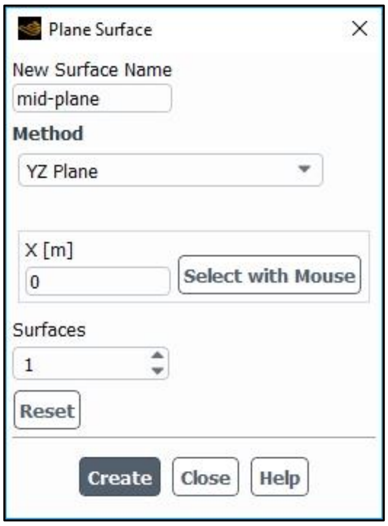


**Figure E1-21.** *Contours* dialog box.

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. E1-22.

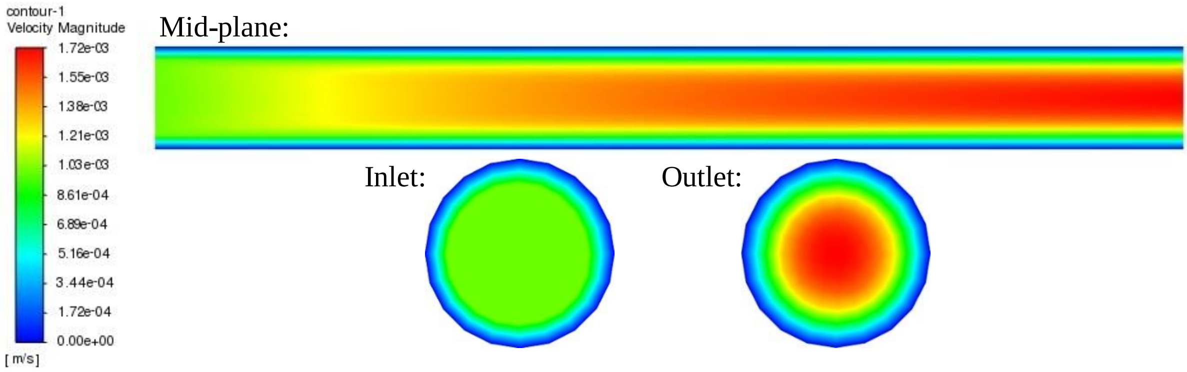


**Figure E1-22.** Contours of fluid velocity magnitude in the mid-plane, at the inflow and outflow surfaces.

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. E1-23a):

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

and confirm by clicking *Create*. In analogous way create two remaining lines with the settings showed in Figs. E1-23b, c.

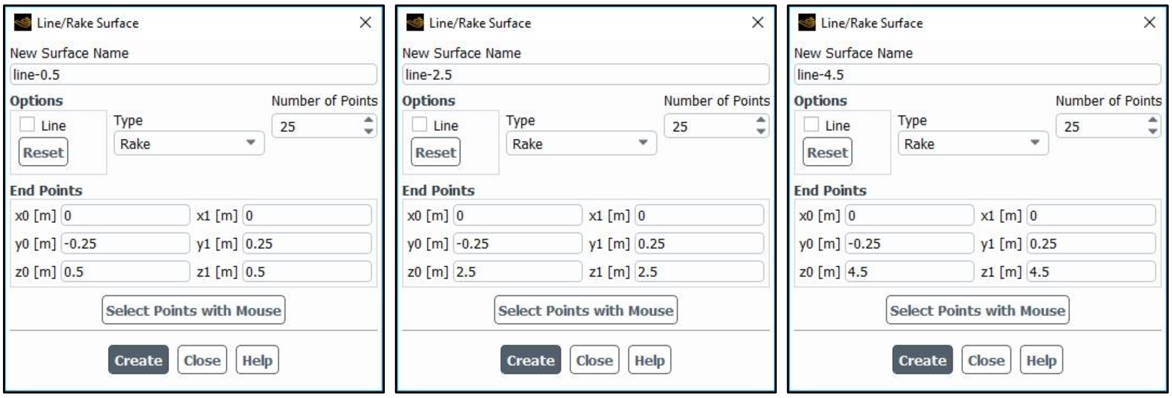


**Figure E1-23.** Settings of the three line locations needed to *XY Plot* creation.

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. E1-24):

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. E1-24).

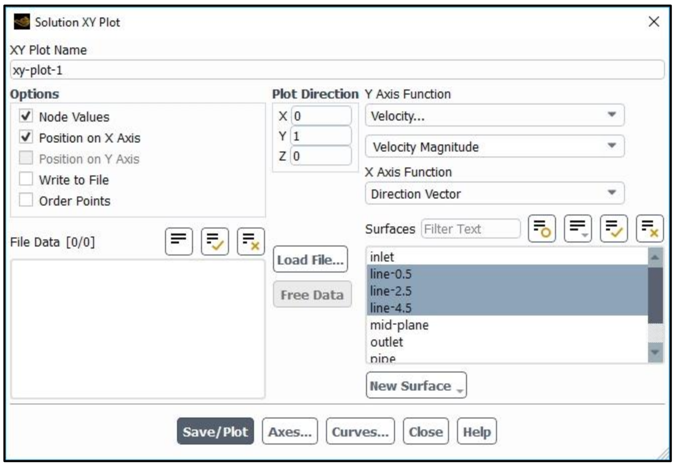


**Figure E1-23.** Settings in the *Solution XY Plot* dialog box.

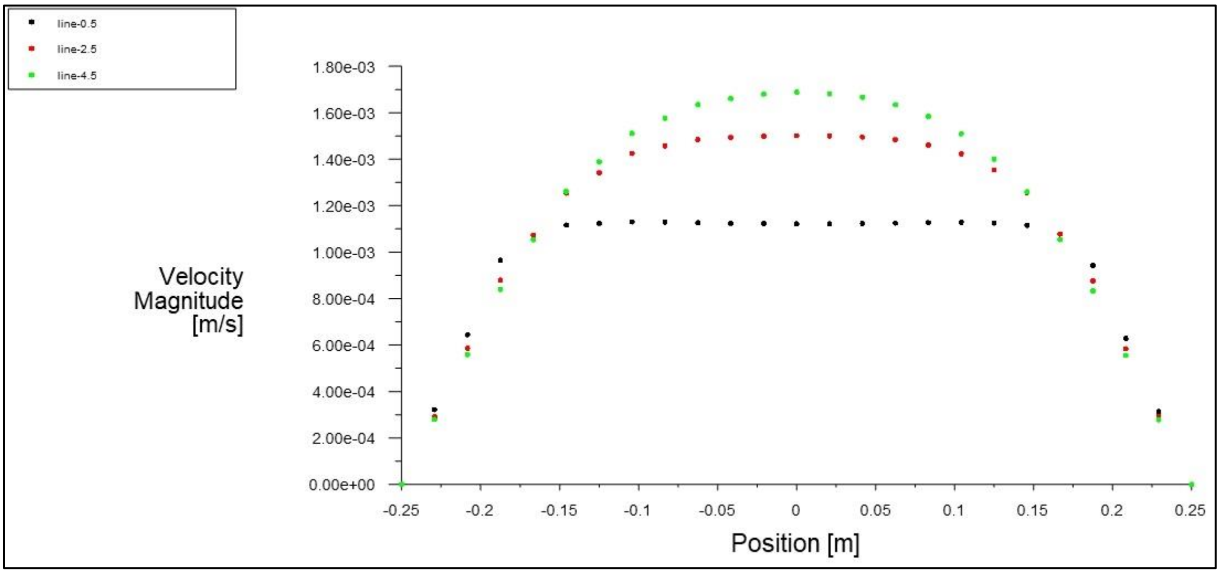


**Figure E1-24.** Profiles of velocity magnitude along the pipe diameter in three locations.