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.

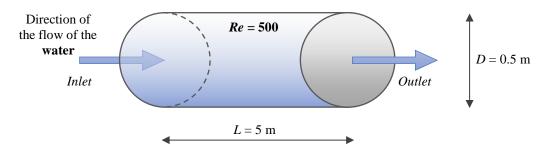


Figure E1-1. Problem description.

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 $Create \rightarrow Primitives \rightarrow Cylinder$ (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 \rightarrow Save Project. You can close Ansys DesignModeler.

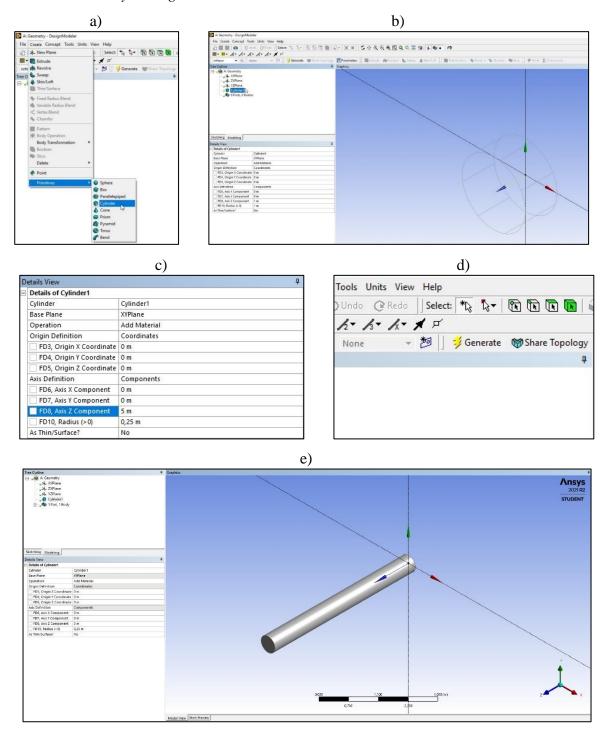


Figure E1-2. Steps of the geometry drawing.

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*.

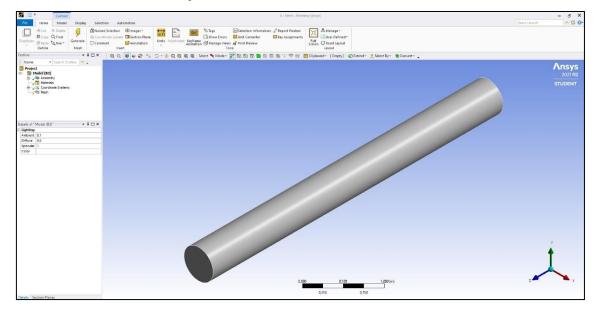


Figure E1-3. Ansys Meshing basic view.

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* \rightarrow *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).

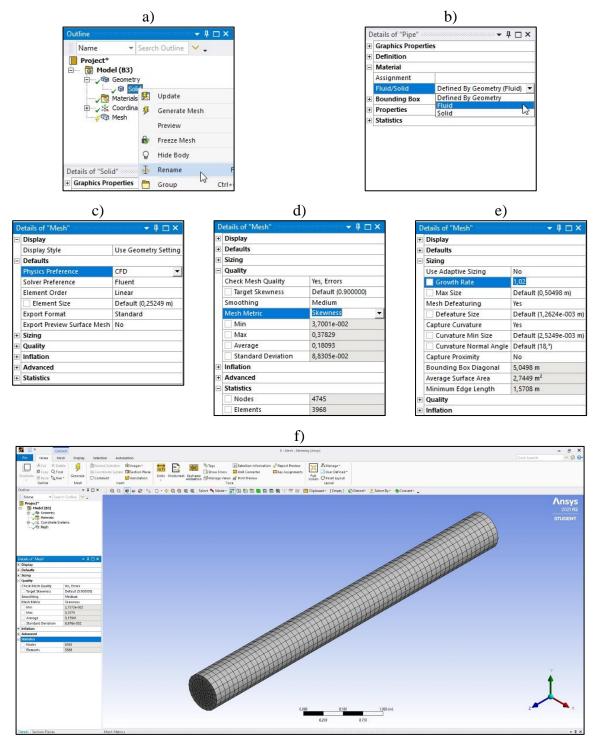
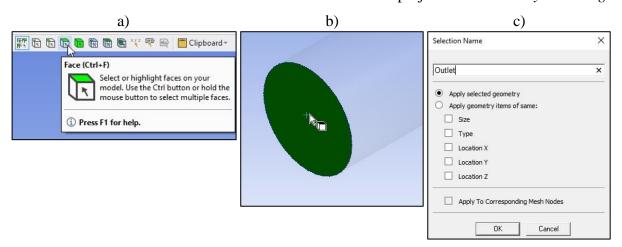


Figure E1-4. Steps of the mesh generation.

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.



d)

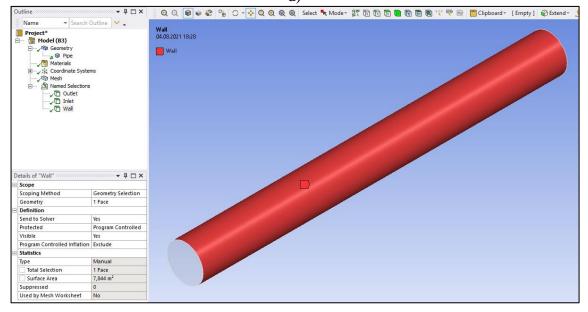


Figure E1-5. Creation of the Named Selections.

1.5. Simulations

Open the Ansys Fluent software by a doubleclick 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.

general-purpose	setup, solve, an	and transient industrial applicatior d post-processing capabilities of A ls for multiphase, combustion, ele	NSYS FI	luent
		Dimension		
		🔘 2D		
		③ 3D		
		Options		
		Double Precision		
		Display Mesh After Read	ing	
		Do not show this panel a	igain	
		Load ACT		
		Parallel (Local Machine)		
		Solver Processes	1	4

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.

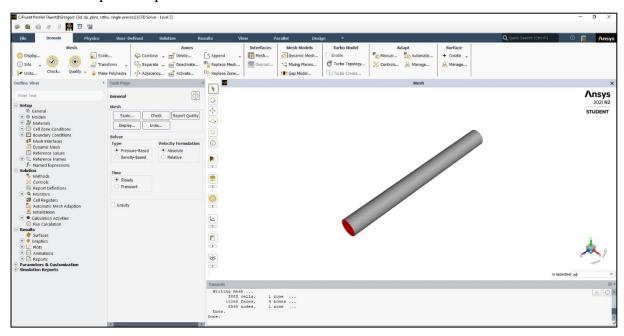


Figure E1-7. Ansys Fluent layout.

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:

- 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* \rightarrow *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.

General Mesh	(?)
Scale C	heck Report Quality
Display Ur	nits
Solver	
Туре	Velocity Formulation
 Pressure-Based Density-Based 	Absolute Relative
Time	
 Steady Transient 	

Figure E1-8. General Task Page.

- Solver Type → Pressure-Based proper for low-speed incompressible to high-speed compressible flows.
- Solver Velocity Formulation → Absolute proper for cases where majority of the domain is non-rotational.
- 5. *Time* \rightarrow *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* \rightarrow *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.

Task Page	<
Models	?
Models	
Multiphase - Off	
Energy - Off	
Viscous - Laminar	
Radiation - Off	
Heat Exchanger - Off	
Species - Off	
Discrete Phase - Off	
Solidification & Melting - Off	
Acoustics - Off	
Structure - Off	

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.

fask Page	
Materials	(?)
Materials	
Fluid	
air	
Solid aluminum	
aiuminum	
Create/Edit Delete	

Figure E1-10. Materials Task Page.

Create/Edit Materials				×	Fluent Database Materials			×
Name air Chemical Formula	Material Type Fluid Fluent Fluid Materials air Mixture none	*	Order Materia Name Chemical F Ident Da GRANTA MDS User-Defined	ormula Itabase 5 Database	Fluent Fluid Materials [1/565] vinyl-trichlorosilane (sicl3ch2ch) vinyldene-chloride (ch2cl2) water-viapor (h2c) water-viapor (h2c) wood-volatiles (wood_vol) Copy Materials from Case Delete		Material Type fluid Order Materials by Name Chemical Formu	
Properties Density [kg/m²] Viscosity [kg/(m s)]	225	•	Edit		Cp (Specific Heat) [J/(kg K)]	998.2	*	View
	nange/Create] Delete] [Close] [Holp	1			Viscosity [kg/(m s)]	0.6 constant 0.001003	v	View

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.

Task Page	Fluid							×
Cell Zone Conditions	Zone Name pipe							
Zone Filter Text	Material Name air	₹) Edit.						
pipe Phase Type ID	Frame Motion Forous Zone	uid Consource						
mixture V fluid V 2	Reference Frame Mesh	Motion Porous Zone	3D Fan Zone	Embedded LES	Reaction	Source Terms	Fixed Values	Multiphase
Edit Copy Profiles	Rotation-Axis Origin			Rotation-Axis	Direction			
Parameters	× [m] 0		•	XO				*
Display Mesh	Y [m] 0		•	YO				•
	Z [m] 0		•	Z 1				•
Orous Formulation Superficial Velocity Physical Velocity			Apply	Close)			

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).

Task Page		Task Page		
Boundary Conditions	(?)	Boundary Co	nditions	(?)
Zone (Filter Text		Zone Filter Te	ext	
inlet interior-pipe outlet wall		inlet interior-pipe outlet wall		
Phase Type ID		Phase	Туре	ID
mixture		mixture 💌	wall 🔻	7
Edit Copy Profiles		Edit	mass-flow-outlet outflow outlet-vent	files
Parameters		Parameters		ditions
Operating Conditions		Display Mes	pressurr-field pressure-inlet	
Display Mesh Periodic Conditions			pressure-outlet symmetry	itions
			velocity-inlet	IIIs
Perforated Walls			wall	

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 \rightarrow the *interior* boundary condition,
- 2. for the *outlet* zone \rightarrow 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).

nlet									
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	Structure	U	JDS
V	elocity Spe	cification Me	thod Mag	nitude, M	lormal to Bou	indary			*
	F	Reference Fr	ame Abso	lute					¥
	Ve	elocity Magn	itude [m/s	0.0010	05			0	-
	sonic/Initia	Gauge Pres	ssure [Pa]	0					

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 \rightarrow Coupled,
- 2. Flux Type \rightarrow Auto Select,
- 3. Gradient \rightarrow Least Squares Cell Based,
- 4. Pressure \rightarrow Second Order,
- 5. Momentum \rightarrow Second Order Upwind,
- 6. Other options are disabled.

Figure E1-15. Solution Methods Task Page.

Task Page	
Solution Methods	(?
Pressure-Velocity Coupling	
Scheme	
Coupled	-
Flux Type	
Rhie-Chow: momentum based	 Auto Selection
Spatial Discretization	
Gradient	
Least Squares Cell Based	*
Pressure	
Second Order	Ŧ
Momentum	

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.

Options	Equations			
✓ Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria
✓ Plot	continuity		v	1e-13
Curves Axes	x-velocity		•	1e-13
Iterations to Plot	y-velocity	✓	✓	1e-13
1000	z-velocity		v	1e-13
Iterations to Store				
1000				
	Convergence	Conditions		
	Show Advar	nced Options		

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:

- 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.

Task Page		
Solution Initialization Initialization Methods Hybrid Initialization Standard Initialization More Settings Initialize Patch Reset DPM Sources Reset LWF	Reset Statistics	•
Task Page		
Solution Initialization Initialization Methods Hybrid Initialization Standard Initialization	C	
Compute from inlet		
Reference Frame Relative to Cell Zone Absolute		
Initial Values		
Gauge Pressure [Pa]		
X Velocity [m/s]		
Y Velocity [m/s] 0 Z Velocity [m/s]		
0.001005 Initialize Reset Patch		

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*".

	?
Update Dynamic	Mesh
Reporting Interval	
1	\$
ly Statistics	
Quantities	
culate	
	Reporting Interval 1 Vy Statistics Quantities

Figure E1-18. Run Calculation Task Page.

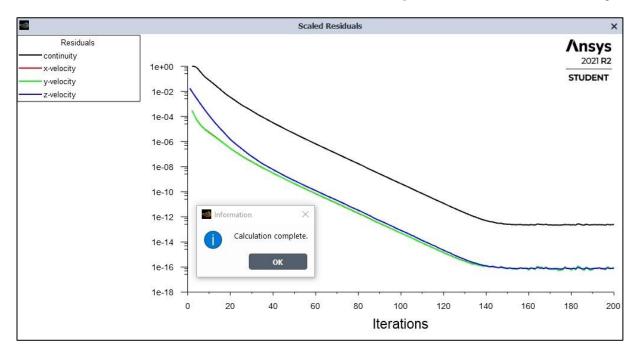


Figure E1-19. Residuals for the converged solution.

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.

In this purpose from the *Contours* dialog box choose *New* Surface \rightarrow Plane... A Plane Surface dialog box will open. Here enter the following settings (Fig. E1-21):

- 1. New Surface Name \rightarrow mid-plane,
- 2. Method \rightarrow YZ Plane,
- 3. $X \rightarrow 0$,
- 4. Surfaces $\rightarrow 1$,

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.

contour-1			
Options	Contours of		
 ✓ Filled ✓ Node Values ✓ Boundary Values ✓ Contour Lines ✓ Global Range ✓ Auto Range 	Velocity		
	Velocity Magnitude		
	Min	Max	
	0	0	
	Surfaces Fil	ter Text 📃 🗾	= -
Clip to Range Draw Profiles Draw Mesh	inlet outlet wall		
Coloring			
 Banded Smooth 			
Colormap Options	Display Sta	te	
	None	• Use Active	New Surface 🖕

Figure E1-20. Contours dialog box.

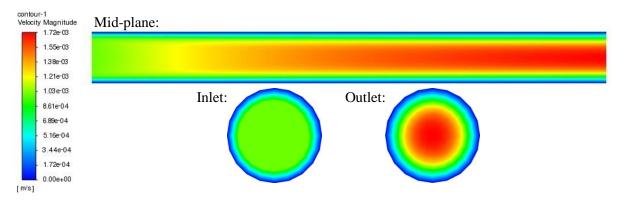
Plane Surface	×
New Surface Name	9
mid-plane	
Method	
YZ Plane	*
X [m]	Select with Mouse
Surfaces	
1	
Reset	
Create	Close
50	

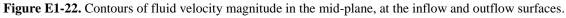
After complete this operation a new plane will appear on the *Surfaces* list. Come back to the *Contours* dialog box and enter the following settings:

- 1. Contours of \rightarrow Velocity... \rightarrow Velocity Magnitude,
- 2. Surfaces \rightarrow mid-plane,
- 3. Coloring \rightarrow 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.





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* \rightarrow *Line/Rake*... A *Line/Rake Surface* dialog box will open. Here enter the following settings (Fig. E1-23a):

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

$$x1 \ [m] = 0, \quad y1 \ [m] = 0.25, \quad z1 \ [m] = 0.5$$

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

Line/Rake Surface	×	Line/Rake S	urface		×	Line/Rake	Surface			×
New Surface Name		New Surface Na	ame			New Surface	Name			
line-0.5)	line-2.5				line-4.5				
Options	Number of Points	Options			Number of Points	Options			Number (of Points
Line Type Reset Rake	• 25 •	Line Reset	Type Rake	*	25	Line Reset	Type Rake	٣	25	*
End Points		End Points				End Points				
x0 [m] 0	x1 [m] 0	x0 [m] 0		x1 [m] 0		x0 [m] 0		x1 [m] 0		
y0 [m] -0.25	y1 [m] 0.25	y0 [m] -0.25		y1 [m] 0.2	25	y0 [m] -0.2	5	y1 [m] 0.3	25	
z0 [m] 0.5	z1 [m] 0.5	z0 [m] 2.5		z1 [m] 2.5	;)	z0 [m] 4.5		z1 [m] 4.	5	
Select Pol	Close Help	(Select Poin	ts with Mou Close He			Select Po	ints with Mou		

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* \rightarrow *X* = 0, *Y* = 1, *Z* = 0,

- 2. YAxis Function \rightarrow Velocity... \rightarrow Velocity Magnitude,
- 3. X Axis Function \rightarrow Direction Vector,
- 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).

xy-plot-1			
Options	Plot Direction Y Axis Funct	tion	
✓ Node Values	X 0 Velocity		*
Position on X Axis	Y 1 Z 0 Velocity Ma	agnitude	•
Write to File	X Axis Fund	tion	
Order Points	Direction V	ector	*
File Data [0/0]	Surfaces Fi Load File Free Data	iter Text	
	New Surfa	ace "	

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

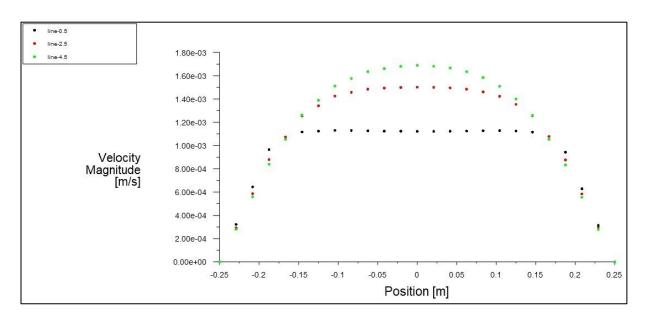


Figure E1-24. Profiles of velocity magnitude along the pipe diameter in three locations.