# $\begin{array}{l} {\rm STAR-CCM+\ Introduction\ Tutorial:} \\ {\rm Laminar\ Flow\ in\ a\ Rectangular\ Duct} \\ {\rm R2.00} \end{array}$

Bronwyn Rempel Ethan Axdal Patrick Gareau Scott J. Ormiston

Department of Mechanical Engineering University of Manitoba Canada

2025-01-08

# Contents

Ι	Creating the Duct Geometry and Mesh	<b>2</b>
1	Introduction for Part I         1.1       Overview         1.2       Objectives         1.3       Saving in Stages	<b>3</b> 3 3 3
2	Creating the Duct Geometry2.1Opening the STAR-CCM+ Software2.2Creating the 3D-CAD Model2.3Creating a Part from a 3D-CAD Model	<b>3</b> 3 5 11
3	Meshing         3.1       Creating a Directed Mesh	<b>16</b> 16
п	Laminar Flow Simulation	<b>21</b>
4	Introduction for Part II         4.1       Overview	<ul> <li>21</li> <li>21</li> <li>21</li> <li>22</li> <li>23</li> </ul>
5	Setting Up the Simulation5.1Starting from the Previous Simulation5.2Selecting the Physics Models5.3Setting the Fluid Properties and Initial Conditions5.4Specify the Boundary Conditions5.5Set the Stopping Criteria	<ul> <li>23</li> <li>23</li> <li>24</li> <li>26</li> <li>27</li> <li>32</li> </ul>
6	Running the Simulation	35
7	Post-Processing the Simulation in <b>STAR-CCM+</b>	36
8	Further Exploration	45

# Part I Creating the Duct Geometry and Mesh

# 1 Introduction for Part I

#### 1.1 Overview

This tutorial covers the steps necessary to create a 3D model of laminar flow in a duct. The content is based on Release 2410 (Version 19.06.008).

In the first part, a simple rectangular duct geometry is created in the STAR-CCM+ 3D-CAD Modeler. The duct that will be created has a length  $L_x = 2.0$  [m], a height  $L_y = 0.2$  [m], and a depth  $L_z = 0.3$  [m]. A uniform orthogonal mesh is then created in the domain using a directed mesh. The second part covers the steps required to set up and run a laminar flow simulation in the duct domain.

#### 1.2 Objectives

This part is intended to demonstrate how to:

- 1. Open STAR-CCM+ using the Simcenter STAR-CCM+ Power on Demand License
- 2. Create a geometry using the STAR-CCM+ 3D-CAD Modeler
- 3. Convert the geometry created using the STAR-CCM+ 3D-CAD Modeler into a part for analysis
- 4. Create a directed mesh
- 5. Save the simulation

#### 1.3 Saving in Stages

It is strongly recommended that you save your work under a different file name(using Save As) after each section in this tutorial. After saving you can re-open the original simulation file and continue. Saving periodically along the way allows you to get back to a previous working stage in case something should not work as expected.

# 2 Creating the Duct Geometry

This section will outline the process of creating the duct geometry using the STAR-CCM+ 3D-CAD Modeler. Note that the created duct geometry will only be half the height of the analyzed duct because symmetry will be assume about the x-z plane at y = 0.

#### 2.1 Opening the **STAR-CCM+** Software

This section will show you how to open a new simulation using the STAR-CCM+ software, including how to set your license, how to input your key, and how to save a file.

- 1. Launch STAR-CCM+.
- 2. At the top of the window, in the *Toolbar*, click on the Create a File icon.
- 3. Ensure the file type is set to Simulation and set the *License* to *Simcenter* STAR-CCM+ Power on Demand.

4. If it is your first time using STAR-CCM+, enter your POD key for the STAR-CCM+ license you have been provided. If this is not your first time using STAR-CCM+, the key should already be entered.

Once this is complete, click OK. This process is highlighted in Figure 1. More information on using Power on Demand Licensing can be found on Page 119 of the User Guide.

Create a File X
Type: Simulation
Template: V Select
Process Options
Serial
O Parallel on Local Host
O Parallel on Named Hosts
O Parallel Specified by Machine File
Remote Server
Remote Host:
Remote User:
Remote Shell:
GPGPU Usage (Linux servers only)
Saved Configurations Microsoft HPC Server
Save Save
License
Simcenter STAR-CCM+ Power on Demand 🗸 Server: 1999@flex.cd-adapco.com
Key:
Server Connection Mode: Default
Rendering: Local V Host: Iocalhost Port: 47927 V Scan
Command: rccm+ -server -power -podkey AntuiXOyVZgmL7ST/VSQ7w -licpath 1999@flex.cd-adapco.com
OK Cancel Help

Figure 1: STAR-CCM+ license selection screen.

5. When you open the STAR-CCM+ software, a layout similar to Figure 2 should appear. This layout will be referenced throughout the tutorial. More information on the STAR-CCM+ menu layout can be found on Page 186 of the User Guide.



Figure 2: STAR-CCM+ layout.

At the top of the window, click on File → Save As...
 Select an appropriate location for the Simulation file and set the file name to duct. Click Save.

#### 2.2 Creating the 3D-CAD Model

This section will summarize the process of creating a model using the STAR-CCM+ *3D-CAD Modeler*, including how to prepare grid lines for sketching, how to create and dimension a sketch, and how to use the extrude feature.

1. In the *Explorer Pane*, expand the Geometry node. Right click on 3D-CAD Models and click New, as shown in Figure 3. The *STAR-CCM+ 3D-CAD Modeler* will open in the *Graphics Window*.



Figure 3: Opening the STAR-CCM+ 3D-CAD Modeler.

2. Right click on the YZ node under Features and select Create a Sketch. Grid lines will appear in the *Graphics Window*.

You can zoom in or out by scrolling the mouse. To rotate the grid, left click in the Graphics Window and drag the mouse. To pan, right click in the Graphics Window and drag the mouse. These mouse commands will work for any scene in the Graphics Window.

3. Under the Display Options click on the View Normal to Sketch Plane icon. This will align the sketch plane normal to the screen. This icon is shown in Figure 4.

Sketch: Sketch 1	
Underlying Sketch Plane: YZ	
Create Sketch Entities	۲
$\begin{array}{c} 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 $	
Sketch Operations	8
Display Options	8

Figure 4: View Normal to Sketch Plane icon.

4. Click on the Set Sketch Grid Spacing icon under the Display Options section. This icon is shown in Figure 5.

Sketch: Sketch 1 Underlying Sketch Plane: YZ	
Create Sketch Entities	۲
1 + 2 + 3 + 3	
Sketch Operations	۲
Display Options	۲

Figure 5: Set Sketch Grid Spacing icon.

5. The *Grid Spacing* dialog box will open. Change the grid spacing to 0.05 [m] and change the number of fine grid deviations to 1.0, as shown in Figure 6. Once this is complete, click OK.

🕞 Grid Spacing						
Grid spacing 0.05 m						
Number of fine grid divisions 1.0						
	ОК	Cancel	Help			

Figure 6: Grid Spacing menu.

You may need to zoom out by scrolling to decrease the size of the grid at this point. You may also need to right click to pan either horizontally or vertically in the Graphics Window.

6. Click on the Rectangle icon in the Create Sketch Entities section, shown in Figure 7.

etch Entities	8
$\begin{array}{c} 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 \\ 1 $	
erations	8
$\times 40$	
tions	6
	etch Entities $ \begin{array}{c} 1 \\ 1 \\ 2 \\ 3 \\ 3 \\ 2 \\ 3 \\ 2 \\ 3 \\ 3 \\ 2 \\ 3 \\ 3 \\ 2 \\ 3 \\ 3 \\ 3 \\ 3 \\ 3 \\ 3 \\ 3 \\ 3 \\ 3 \\ 3$

Figure 7: Rectangle sketch tool.

7. Create a rectangle spanning 4 squares horizontally and 6 squares vertically by clicking on the origin of the sketch plane and again on the opposite corner of the rectangle. The sketch should look similar to Figure 8.



Figure 8: Initial duct cross-section sketch.

The sketch will automatically attempt to lock onto grid lines.

- 8. Right click on one of the horizontal lines and select the Apply Length Dimension under the Dimensions section. The Dimension Window will open. If necessary, change the length of the line to 0.2 [m] and click OK.
- 9. Right click on one of the vertical lines and select the Apply Length Dimension under the Dimensions section. The Dimension Window will open. If necessary, change the length of the line to 0.3 [m] and click OK. The completed sketch with dimensions is shown in the following Figure 9.



Figure 9: Completed duct cross-section sketch.

- 10. Once you have defined the dimensions of your sketch, click on the OK button in the *Explorer Pane* to create the sketch.
- 11. Right click on Sketch 1 and select Rename. Rename the sketch to Duct Cross-Section Sketch. Alternatively, you can use the F2 hotkey to rename a node in the *Explorer Pane*.
- 12. Right click on Duct Cross-Section Sketch and select the Extrude option. This process and the renaming process is shown in Figure 10.

	-				
${\rm duct} \; \times \;$					
Simulati	ion Scene/Plot	3D-CAD			Es
🖗 3D-C	AD Model 1				
- 🖻 B	ody Groups				
🛨 🗎 Fi	eatures				
<u>م</u> – ا	XY				
- 1	YZ				
	ZX	-			
- <del>-</del>	Global Origin	Ē	Rename		
	Lab Coordina	te 🕄	Hide		
	Duct Cross-Se	c 🖹	Show		
- 🖻 D	esign Filters	1	Edit		
i 📄 D	esign Paramete	ers 🛄	Edit Sketch Plane		
			Tags	>	
		th	Import Curves		
		r 💫	Extrude		
			Extrude Cut		
		V	Revolve		

Figure 10: Renaming and extrusion features in tree.

13. The Extrusion Window will open in the Explorer Pane. Change the distance to 2.0 [m] and click OK. This can be seen in Figure 11.

Extrude: Extrude 1 —				
Input Type				
Sketch O Edge	s			
Sketch For Extrusion:	Duct	Cross-Se	ection Sketch	
Body Type:	Solid			~
Method:	Blind			~
Direction of Extrusio	n			
Direction Type: No	rmal			~
Extrusion Options:	One Wa	y		~
Distance:	2.0 m			<del>+</del> ?+
Draft:	None			~
Body Interaction:	Merge			~
Bodies To Interact:	All			~
Preview				
	Ś			

Figure 11: Extrusion of cross-section sketch in explorer window.

- 14. Right click on the Extrude 1 feature and select Rename. Rename the extrusion to Duct. Open the Duct  $\rightarrow$  Body Groups node. Right click on Body 1, select Rename, and rename the feature to Duct Body.
- 15. In the *Display Window*, right-click each individual face of the model and rename it using the following naming convention shown in Table 1. Figure 12 depicts renaming the negative x-face to "Inlet":

Face:	Plane:	Name:
Negative x-Face	x = 0	Inlet
Positive x-Face	$x = L_x$	Outlet
Negative y-Face	y = 0	Symmetry Plane
Positive y-Face	$y = L_y$	Top Wall
Negative z-Face	z = 0	Left Wall
Positive z-Face	$z = L_z$	Right Wall

Table 1: Naming Conventions for Duct Faces



Figure 12: Renaming for faces in the 3D-CAD modeler.

16. Click on the Close 3D-CAD button in the *Explorer Pane*. The CAD model is now complete.

#### 2.3 Creating a Part from a 3D-CAD Model

Now that the 3D-CAD model is complete, it must be converted to a geometry part for meshing and analysis. This section summarizes the process of creating a part from a 3D-CAD Model in STAR-CCM+. Alternatively, one could have created a 3D-CAD model in another software, imported it, and then it may be converted into a part using the same process.

- 1. Click on Geometry  $\rightarrow$  3D-CAD Models and rename 3D-CAD Model 1 to Duct.
- 2. To create a part from the 3D-CAD Model in STAR-CCM+, right click on Geometry  $\rightarrow$  3D-CAD Models  $\rightarrow$  Duct. Select the New Geometry Part option, as shown in Figure 13.



Figure 13: Process of creating a part from a 3D-CAD model.

3. The Part Creation Options dialog box will open. For this tutorial, the default settings will be what is used, so click OK. This process is shown in the following Figure 14:

	Create Parts From CAD Solids			
	Create Parts From CAD Sheets			
Part Type	CAD Part with 3D-CAD Association			
	Create Part Contacts from Coincident Entitie			
Coincidence Tolerance	1.0E-5			
Faceted Bodies				
Create Parts From Faceted Solids				
Create Parts From Faceted Sheets				
Part Type Mesh Part with 3D-CAD Association				
Common Options				
	Create Shells From Sheets			
Mark Feature Edge	s Sharp CAD edges			
Sharp Edge Angle (deg	) 30.0			
Tessellation Density	y Medium			

Figure 14: Part creation options dialog box.

4. To view your geometry in the *Display Window*, right click on Scenes and select New Scene  $\rightarrow$  Geometry. This is shown in the following Figure 15.

			Ee.
b duct			
🗕 Geom	netry		
÷ 🗎 30	-CAD Models		
► ©	Duct		
🕨 🚞 Pa	rts		
🕨 🚞 De	scriptions		
- 🖻 Co	intacts		
🗖 🖻 Op	perations		
🗆 🚞 Conti	nua		
🖻 🗎 🗎	ns		
🖻 🗎 Deriv	ed Parts		
📄 Stopp	oing Criteria		
📄 📄 Solut	ion Histories		
🕨 🚞 Solut	ion Views		
🕨 📄 Moni	tors		
Plots			
Contraction of the second			
Scene	New Scene	>	Geometry
	New Scene Open All Scenes	>	Geometry Mesh
C S C C C C C C C C C C C C C C C C	New Scene Open All Scenes Apply Representation	on >	Geometry Mesh Scalar
···· (a) Strand ···· (a) L ··· (a) St (a) ··· (a) R ··· (a) T(	New Scene Open All Scenes Apply Representati Test Graphics	> on >	Geometry Mesh Scalar Vector
	New Scene Open All Scenes Apply Representation Test Graphics Edit	on >	Geometry Mesh Scalar Vector Empty
	New Scene     Open All Scenes     Apply Representation     Test Graphics     Edit     New Group	on >	Geometry Mesh Scalar Vector Empty
S	New Scene Open All Scenes Apply Representation Test Graphics Edit New Group UnGroup	> on >	Geometry Mesh Scalar Vector Empty
	New Scene       Open All Scenes       Apply Representation       Test Graphics       Edit       New Group       UnGroup       Paste	on >	Geometry Mesh Scalar Vector Empty

Figure 15: Extrusion of cross-section sketch in explorer window.

5. The duct should now be available in the *Display Window* and should look similar to Figure 16.



Figure 16: View of duct body in geometry scene.

6. Prior to setting up the mesh for the geometry, the geometry part must be converted into regions. Regions are various surfaces on the part where you can set boundary conditions and define the behaviour of the surfaces of a part. To do this, right click on Geometry  $\rightarrow$  Parts  $\rightarrow$  Duct Body and select Assign Parts to Regions...

This process is shown in Figure 17.



Figure 17: Process of assigning parts to regions.

- 7. The Assign Parts to Regions dialog will open. Select the Duct Body as the part to assign to regions. Change the pull down menus as required to read the following:
  - Create a Region for Each Part
  - Create a Boundary for Each Part Surface
  - Create Boundary-mode Interfaces From Contacts

The selected options on the pull down menus should look like Figure 18. Once this is completed click Apply and Close.

Assign Parts to Regions	×
🖓 🔚 🔚 🕒 🕵 🔍 Filter by Path	.* 🗉 🗆
Parts	
— I Duct Body	
1 of 1 selected	
Create a Region for Each Part 🔹	Select
Create a Boundary for Each Part Surface 🔻	Select
Create Boundary-mode Interfaces From Contacts	
Apply Close	Help

Figure 18: Assign Parts to Regions menu selections.

8. To ensure you have correctly assigned parts to regions correctly, you can expand the Regions  $\rightarrow$  Duct Body  $\rightarrow$  Boundaries node. Each face named when creating the 3D-CAD model should have been converted into a separate boundary as seen in Figure 19:



Figure 19: Boundaries created when assigning parts to regions.

9. The part geometry has been created and set up for meshing. Ensure the simulation is saved prior to starting the meshing section by clicking File  $\rightarrow$  Save, and title the simulation duct.sim.

# 3 Meshing

This section outlines the process of creating a mesh for a given part. The purpose of creating a mesh is to divide the geometry into volumes (or areas for a 2D mesh) used in the discretisation process. This section will also summarize the procedure of exporting a mesh for use in later simulations. The saved file can be used in the subsequent tutorial: *STAR-CCM+ Introduction Tutorial Part 2: Laminar Flow in a Rectangular Duct.* The meshing process used is just one type of mesh generation available in STAR-CCM+.

#### 3.1 Creating a Directed Mesh

The type of mesh that will be used in this analysis will be a directed mesh. The key features of a directed mesh are the source surface and the target surface. A patch mesh is created on the source surface and is extended through the use of a volume distribution to the target surface.

1. To begin creating the mesh for the simulation, right click on Geometry  $\rightarrow$  Operations and select New  $\rightarrow$  Mesh  $\rightarrow$  Directed Mesh, as shown in Figure 20.

duct ×		-	Geo	metry Scene 1 ×
Simulation Scene/Pl	ot	°a  ▼	si	mcenter STAR-CCM+
😽 duct			5.	incenter SIAR cent
🔻 🗎 Geometry				
🕨 🗎 3D-CAD M	odels			
🔻 📄 Parts				
► 🕼 Duct Bo	ody			
Descriptio	ns			
Contacts	-			
📄 📄 Opera	New >	Surface Preparati	on >	
Continua	Execute All	Boolean	>	
Derived F		Mesh	>	Automated Mesh
Stopping	Reorder	incon		Radge for 2D Meshing
Solution	Paste Ctrl+V			Automated Mach (2D)
Solution	Refresh			Automated Mesh (2D)
Reports				Automated Mesh (Shen)
Monitors				Volume Extruder
🕨 🚞 Plots				Directed Mesh
🕶 🚞 Scenes			[	Volume Mesh Pattern
🕨 🖴 Geometry	Scene 1			
📄 🗎 Layout Views				Ŷ
– 🗎 Summaries				L_x ×
🕨 🗎 Representatio	ons			<b>a</b>
🕨 🚞 Tools				

Figure 20: Process of selecting the directed mesh option.

- 2. The Create Directed Mesh Operations menu will open on screen. Click on Duct Body to select the body you wish to create the mesh for and click OK. A new Directed Mesh node will appear under Operations in the *Explorer Pane*.
- 3. Right click on Geometry  $\rightarrow$  Operations  $\rightarrow$  Directed Mesh and select Edit... This process is shown in Figure 21. The Directed Mesh window will open in the *Explorer Pane*.

duct ×		
Simulation Scene/Plot		Ê
😽 duct		
🔻 🚞 Geometry		
🕨 📄 3D-CAD Model	s	
🔻 🚞 Parts		
🕨 🕼 Duct Body		
🕨 🚞 Descriptions		
Contacts		
🛨 📄 Operations		
🕨 🔏 Directed M	e l'a	
🗎 Continua	Edit	
🕨 🖻 Regions	Execute	
📄 Derived Parts	Repair Input Surf	ace
📄 Stopping Crite	Tags	>
🗎 Solution Histo	Conv	Ctrl+C
Solution Views	Сору	Ctrl+C
🗎 🗎 Reports	Paste	Ctrl+V
Monitors	Delete	Delete
🕨 📄 Plots	Rename	

Figure 21: Process of opening the mesh editor.

4. Select the Source Surfaces node and in the *Properties Window*, open the Custom Editor and select the Inlet as the source surface. The button to open the Custom Editor is shown in Figure 22.

Source Surfaces - Prop	erties ×	_
<ul> <li>Properties</li> </ul>		
Source Surfaces	[]	¥

Figure 22: Button to open the Custom Editor.

5. Select the Target Surfaces node and in the *Properties Window*, open the Custom Editor and select the Outlet as the target surface.

The Directed Mesh will begin on the Source Surface and travel to the Target Surface, which is why these surfaces must be defined prior to creating the mesh.

- 6. Right click on the Source Meshes node and select New Source Mesh  $\rightarrow$  Patch Mesh. The Part Collection For This Source Mesh window will open in screen. Select the Duct Body as the part you wish to create the source mesh for and click OK.
- 7. The Patch Mesh Editor will open, which is where the mesh is created on the source surface. Click on the Auto-Populate Feature Edges with Patch Curves icon under the Create/Edit Patch Entities section. This icon is shown in Figure 23. Once this is done correctly, the edges of the source surface will change colour from yellow to green. Your patch topology should begin by looking like Figure 24a and end looking like Figure 24b.



Figure 23: Auto-Populate Feature Edges with Patch Curves icon.



Figure 24: Patch topology on source surface.

The Auto-Populate Feature Edges with Patch Curves feature will automatically create one large patch mesh around the perimeter of the source surface. This is the desired patch mesh for the simulation, so the patch mesh creation is complete. 8. At the top of the *Explorer Pane*, change to mode from Patch Topology to Patch Mesh. This is where the number of nodes in each direction on the source surface are specified. Click on one of the horizontal edges. The Mesh Properties section in the *Explorer Pane* will become available once done correctly. Change the number of divisions to 15 and click Apply, as shown in Figure 25.

luct × -	Geometry Scene 1 × Directed Mesh 1 ×
imulation ScenerPlot Directed Mesh Edit 🖁 🖣	Simcenter STAR-CCM+
Mesh Properties	
Number of Divisions : 15	
Distribution Type: Constant Reverse Direction : Spacing at Start: 0.1 m Spacing at End: 0.1 m	
Smooth Number of iterations: Apply Distribution after Smoothing: Apply Distribution after Smoothing:	į. Ka

Figure 25: Process of specifying the horizontal mesh divisions.

9. Click on one of the vertical edges. The Mesh Properties section in the *Explorer Pane* will become available once done correctly. Change the number of divisions to 10 and click Apply, as shown in Figure 26.

duct ×	- Geometry Scene 1 × Directed Mesh 1 ×
Simulation Scene/Plot Directed Mesh Edit	Simcenter STAR-CCM+
Mode Patch Mesh 🛛 🗸	
Mesh Properties	
Number of Divisions : 10	
Distribution	
Type : Constant ~	
Reverse Direction :	
Spacing at Start: 0.1 m	
Spacing at End: 0.1 m	
Apply	
Smooth	
Number of Iterations : 0	•
Apply Distribution after Smoothing :	ť.
Apply	44

Figure 26: Process of specifying the vertical mesh divisions.

10. Once both actions are completed correctly, the mesh on the source surface should appear in the *Display Window*. When the mesh looks similar to the mesh in Figure 27, click on Close to close the surface mesh.

			-
		_	
		-	_
		+	-

x z

Figure 27: Patch mesh of source surface.

11. Right click on Mesh Distributions and select New Volume Distribution, as shown in Figure 28. The Parts for Creating a New Mesh Distribution menu will open on screen. Select the Duct Body as

the body to create the volume mesh for and click OK. A new Volume Distribution node will be added under Mesh Distributions. Ignore the Default Distribution node, which appears in the latest software version.

Plot Directed Mesh	
aces	
ces	
arts	
У	
nes	
sh	
outions	
New Volume Dist	ribution
Paste	Ctrl+V
	Plot Directed Mesh ices ces arts y tes sh Nutions New Volume Dist Paste

Figure 28: Creating the volume distribution node.

The purpose of the volume distribution is to set the number of divisions of the patch mesh from the source surface through the body to the target surface.

12. Select Mesh Distributions  $\rightarrow$  Volume Distribution  $\rightarrow$  Default Controls  $\rightarrow$  Number of Layers and in the Properties Window, change the Number of Layers to 40 and hit Enter. This process is shown in Figure 29.



Figure 29: Process defining the volume distribution.

- 13. The directed mesh is now complete, so click on Close Directed Mesh. This will cause STAR-CCM+ to return to the Geometry Scene.
- 14. To update the directed mesh, right click on Geometry  $\rightarrow$  Operations  $\rightarrow$  Directed Mesh and select Execute. The small caution icon  $\overset{\otimes}{=}$  beside Directed Mesh node will vanish once completed  $\overset{\otimes}{=}$ .
- 15. To view the directed mesh, right click on Scenes and select New Scene  $\rightarrow$  Mesh A new scene should open in the *Graphics Window* which shows the Duct Body and the directed mesh, similar to Figure 30.



Figure 30: View of duct body in mesh scene.

16. Save the simulation by clicking File  $\rightarrow$  Save.

# Part II Laminar Flow Simulation

# 4 Introduction for Part II

#### 4.1 Overview

The starting point for this part is the simulation file from Part I. The main objective of this part is to familiarize a new user with the StarCCM+ workflow and interface.

#### 4.2 Geometry Nomenclature

The rectangular duct domain created in **Part I** is shown schematically in Figure 31.



Figure 31: Rectangular Duct Geometry

The duct has length  $L_x$ , height  $L_y$ , and depth  $L_z$ . The flow is assumed to be symmetric about an x-z plane that bisects the duct, and therefore only half of the full duct is modelled. One corner of the duct is defined to lie at the origin, (0,0,0). All faces of the domain were defined during the modelling process and are named as follows: the plane described by x = 0 is denoted as Inlet,  $x = L_x$  as Outlet, y = 0 as Symmetry Plane,  $y = L_y$  as Top Wall, z = 0 as Right Wall, and  $z = L_z$  as Left Wall. Pertinent geometric parameters are given in Table 2.

#### 4.3 Problem Definition

In this tutorial, an isothermal, incompressible, constant property flow of water through a rectangular duct is modelled. There is no heat transfer model engaged; alternative approaches such as using an isothermal flow model or adding heat transfer could be explored as extensions to this tutorial.

The flow is modelled using a rectilinear uniform grid for half the domain, using symmetry in the y direction. All physical parameters required for the simulation are given in Table 2 below.

Parameter	Symbol	Value	Units	Comments
mass flow rate	ṁ	$1.981 \times 10^{-2}$	kg/s	half of the total mass flow of $3.962{\times}10^{-2}~{\rm kg/s}$
density	ρ	997.0	$\rm kg/m^3$	
dynamic viscosity	μ	$8.899 \times 10^{-4}$	kg/m·s	
duct length	$L_x$	2	m	
duct height (half)	$L_y$	0.2	m	half of the total duct height of 0.4 m
duct depth	$L_z$	0.3	m	
hydraulic diameter	$D_h$	0.34286	m	used to calculate the Reynolds number
Reynolds Number	$\operatorname{Re}_{D_h}$	127.2	_	indicates a laminar flow regime

 Table 2: Geometric and Physical Parameters

#### 4.4 Objectives

This part is intended to demonstrate how to:

- 1. Select physics models
- 2. Review and apply initial conditions and understand their importance in solution convergence
- 3. Specify boundary conditions
- 4. Set stopping criteria
- 5. Run a simulation
- 6. Perform some post-processing of results within the  $\mathsf{STAR-CCM}+$  environment
- 7. Export solution data for further post-processing

# 5 Setting Up the Simulation

STAR-CCM+ solves physics problems using a computational mesh. To provide this mesh, you can either (a) create a geometric model within the STAR-CCM+ environment and then use that model to create the mesh within STAR-CCM+, or (b) import an existing mesh. It is strongly recommended to use method (a), which corresponds to continuing with the duct.sim file that you saved from Part I.

It is recommended that you save the simulation periodically during the set-up process. Use Save As to save progress under other file names to allow getting back to a previous stage should something not work as expected.

### 5.1 Starting from the Previous Simulation

- 1. Launch STAR-CCM+.
- 2. Open the previously created duct.sim file.
- 3. Save the simulation as duct-lam.sim in the file directory you created in **Part I**. Close the duct.sim simulation tab.

#### 5.2 Selecting the Physics Models

A physics continuum defines a set of physics models and their associated material. A physics model is associated with one or more regions. With a mesh defined, a region now exists to which a physics model can be applied.

1. Expand the Continua node and right-click on Physics 1 (this is the physics continuum automatically created and assigned to the Duct Body region).



Figure 32: Select Model menu item in Continua

2. Click on Select Models.... The *Physics Model Selection* dialog should appear:

Physics 1	Material	7	Enabled Models
	◯ Gas		Three Dimensional
	⊖ Liquid		
	⊖ Solid		
	O Multi-Component Gas	<optional></optional>	
	O Multi-Component Liquid		
	O Multi-Component Solid		
	○ Multiphase		
	Time	7	
	O Explicit Unsteady		
	O Harmonic Balance		
	O Implicit Unsteady	<optional></optional>	
	O PISO Unsteady		
	○ Steady		
	Optional Models	-	
	Adaptive Mesh		
	Casting		
	Cell Quality Remediation		
	External Continuum	<optional></optional>	
	Mesh Deformation		
	Radiation		
	Solution Interpolation		
	Auto-select recommended	models	

Figure 33: Physics Model Selection dialog box

3. Select the following physics models, in order:

Group Box	Model
Enabled Models	Three Dimensional (pre-selected)
Material	Liquid
Flow	Segregated Flow
	Gradients (selected automatically with above)
Equation of State	Constant Density
Time	Steady
Viscous Regime	Laminar

Table 3: S	ummary o	f physics	models to	select
------------	----------	-----------	-----------	--------

Then press **Close** to exit the dialog box, as no optional models are required for this simulation. Notice the following:

- Since Auto-select recommended models is engaged, the *Physics Model Selection* dialog guides you through the model selection process by prompting you for required models based on previous selections, and selecting certain default models automatically as you make choices. For example, when a continuum has contains a liquid or a gas, it also needs a flow model. Once it has a flow model, it needs a viscous model. If turbulence is activated in a fluid continuum, it will need a turbulence model.
- Water is the default substance defined for a liquid continuum. This can be seen by expanding Continua  $\rightarrow$  Physics 1  $\rightarrow$  Models  $\rightarrow$  Liquid  $\rightarrow$  H20.
- To reverse part or all of the model selection processes, simply clear the checkboxes of the models you want to deactivate. Active models that are grayed out require other models to be deactivated first.
- The color of the **Physics 1** node has turned from gray to blue, indicating that required models have been activated.
- 4. Open the Continua  $\rightarrow$  Physics  $1 \rightarrow$  Reference Values node and select Reference Pressure. Ensure that the Value property is set to to 101325.0 Pa.
  - Flow simulations require the specification of a reference pressure. The reference pressure is simply a device that is used to reduce the numerical roundoff error in the numerical calculations involving pressure. This is necessary since the differences in pressure are important, and these differences can be small relative to the absolute value of the pressure. Other reference values may be required in more complex flow simulations.
  - A short explanation is given in the *Properties Window* when a Property is selected. As well,
     (Help) or F1 can be used to find the entry in the *STAR-CCM+ User Manual* to learn more about most objects.

duct-lam ×		-
Simulation Scene/Plot	ťe	•
😽 duct-lam		1
🕨 🚞 Geometry		
🕶 📄 Continua		
🕶 🥯 Physics 1		
🕨 🚞 Models		
🔻 🚞 Reference Values		
<ul> <li>Maximum Allowable Absolute Pressure</li> </ul>		
Minimum Allowable Absolute Pressure		
Reference Pressure		
Initial Conditions		
🖿 🧰 Regions		
🗝 Derived Parts		
🖿 🧰 Solvers		
🖿 🧰 Stopping Criteria		
🗝 🖻 Solution Histories		
🖻 🛅 Solution Views		
🗝 🖻 Reports		
🕨 🚞 Monitors		•
Reference Pressure - Properties ×		-
<ul> <li>Properties</li> </ul>		
Value 101325.0 Pa		
Reference Pressure		C
The absolute pressure value relative to which all other pre	ssure	s
(gauge) are defined		

Figure 34: Specifying the reference pressure

#### 5.3 Setting the Fluid Properties and Initial Conditions

- 1. Expand the Continua  $\rightarrow$  Physics  $1 \rightarrow$  Models  $\rightarrow$  Liquid  $\rightarrow$  H20  $\rightarrow$  Material Properties Density node and and select the Constant Node.
- 2. Set the Value property to that given for density in Table 2 (enter 997.0).



Figure 35: Entering the fluid density

3. Similarly, set the Value property for Dynamic Viscosity to that given in Table 2 (enter 8.889E-04).

- 4. Expand the Physics  $1 \rightarrow$  Initial Conditions node and select the Pressure node. Verify that the Method is Constant and the Value is 0.0 [Pa]. This defines the initial pressure field of the fluid at the start of the simulation.
- 5. Similarly, select the Velocity node and verify that the Method is constant and that the value is [0.0, 0.0,0.0] [m/s]. This defines the initial velocity field of the fluid as stationary in the Laboratory coordinate system, which is the default reference frame of any continuum.



Figure 36: Specifying the initial conditions

#### 5.4 Specify the Boundary Conditions

Boundary conditions are applied under the Regions  $\rightarrow$  Boundaries node. In Part I, you Assigned Parts to Regions when creating the mesh. This step created Boundaries from previously defined part surfaces. The following boundary conditions are applied to those boundaries:

Boundary	Condition
Inlet	Mass Flow Inlet
Left Wall	Wall
Outlet	Pressure Outlet
Right Wall	Wall
Symmetry Plane	Symmetry Plane
Top Wall	Wall

Table 4:	List of	boundary	$\operatorname{conditions}$
----------	---------	----------	-----------------------------

- 1. Check that the boundaries are properly defined: with Mesh Scene 1 open in the *Graphics Window*, open the Regions  $\rightarrow$  Duct Body  $\rightarrow$  Boundaries node and click on each boundary in sequence. The selected boundary should be highlighted in pink in the graphics window. As well, ensure all boundary conditions are set in the default Laboratory (or Lab) coordinate system.
- 2. Set the **Inlet** boundary condition:

2.1. Select the Inlet node and set the Type property to Mass Flow Inlet.



Figure 37: Inlet boundary condition selection

Note here that the Part Surfaces property contains [Duct Body.Inlet]. The Part Surfaces property dictates the geometric entities (in this case a region) associated with this Boundary.

2.2. Expand the Inlet  $\rightarrow$  Physics Conditions node and select Flow Direction Specification. The Method should be set to Boundary-Normal, but note the other options available.

duct-iam ×		
Simulation Scene/Plot		te l
😽 duct-lam		
🕨 🚞 Geometry		
🕨 🚞 Continua		
🛨 🚞 Regions		
🕶 🔜 Duct Body		
🕶 🚞 Boundaries		
🛨 🎇 Inlet		
👻 🖻 Physics Condit	ions	
- 😾 Flow Direct	ion Specification	
😽 Mass Flow	Option	
😽 Reference F	rame Specification	
Physics Values		
🕨 🚃 Left Wall		
🕨 🐖 Outlet		
🕨 🐖 Right Wall		
🕨 🐖 Symmetry Plane		
🕨 🐖 Top Wall		
Physics Conditions		
Physics Values		
🖻 Derived Parts		
Solvers		
Flow Direction Specification - P	roperties ×	
<ul> <li>Properties</li> </ul>		
Method	Boundary-Normal	•
	Boundary-Normal	
	Components	
	Angles	
Method		6

Figure 38: Mass flow direction specification

2.3. Select the Mass Flow Option node and verify that Specification Option is set to Mass Flow Rate. The alternative option is to define a mass flux at the boundary.

duct-lam ×			_
Simulation Scene/Plot		te I	•
₩ duct-lam			^
🕨 🖻 Geometry			
🕨 🖻 Continua			
🛨 🖻 Regions			
🗕 🗾 Duct Body			
👻 🖻 Boundaries			
🛨 💓 Inlet			
Physics Conditions			
Flow Direction Specification	on		
Mass Flow Option			
😽 😾 Reference Frame Specifica	tion		
Physics Values			
🕨 🐖 Left Wall			
🕨 🐖 Outlet			
🕨 🐖 Right Wall			
🕨 🐖 Symmetry Plane			
🕨 🐖 Top Wall			
Physics Conditions			
Physics Values			
Corived Parts			~
			1
Mass Flow Option - Properties ×			_
▼ Properties			
Specification Option	Mass Flow Rate		Ŧ
	Mass Flux		
	Mass Flow Rate		
Specification Option		(	?
Mass flow/flux specification option			

Figure 39: Mass flow specification option

2.4. Expand the Inlet  $\rightarrow$  Physics Values node and select Mass Flow Rate. Set the Value property

to that given in Table 2 (enter 1.981E-02).

- 3. Set the **Outlet** boundary condition:
  - 3.1. Select the Outlet node and set the Type property to Pressure Outlet.
  - 3.2. Expand the Outlet  $\rightarrow$  Physics Conditions node and select Backflow Specification Option. Change the Pressure property from Environmental to Static. The Environmental condition subtracts the dynamic head at the pressure boundary, while the Static condition maintains the pressure at the user-defined pressure.



Figure 40: Backflow specification

Leave the other conditions as default, noting:

- The Direction property controls the flow direction at the boundary, with options of:
  - Boundary Normal as the name suggests, the flow is normal to the boundary.
  - $\mathsf{Extrapolated}$   $\mathsf{extrapolates}$  the flow from the interior of the domain.
  - Components or Angles these options specify the flow directly in 3D space through either a unit vector or Euler angles.
- The Scalars property controls how the scalar quantities (e.g., temperature) are defined at the boundary, with options of:
  - Specified applies the scalar conditions that you specify if the flow reverses at the boundary, otherwise, extrapolates scalar conditions from the domain.

- $\mathsf{Extrapolated}$  <code>extrapolates</code> the conditions from the domain regardless of the flow direction.
- 3.3. In the same node, select Pressure Outlet Option. Set the Option to Average Pressure.
- 3.4. Expand the Outlet → Physics Values node and select Average Pressure. Verify that the Value property is 0.0 [Pa]. Recall that this is a gauge pressure to the Reference Pressure set in the Physics 1 continua. Therefore, the absolute average pressure at the outlet is 101325.0 [Pa].
- 3.5. Next, select the Average Pressure Adjuster node and change the Blending Factor property to 0.05.



Figure 41: Average pressure adjuster

- 4. Set the Symmetry Plane boundary condition: select the Symmetry Plane node and set the Type property to Symmetry Plane.
- 5. Set the Wall boundary conditions:
  - 5.1. Expand the Left Wall node and verify that the Type property is set to wall.
  - 5.2. Expand the Left Wall  $\rightarrow$  Physics Conditions node and select Shear Stress Specification. Ensure that the Method property is set to No-Slip.

		-	
Simulation Scene/Plot		Ľ:	1
🕨 🚞 Geometry			1
🕨 🚞 Continua			
🕈 🚞 Regions			
🛨 🧱 Duct Body			
👻 🚞 Boundaries			
🕨 🌺 Inlet			
🛨 🐖 Left Wall			
<ul> <li>Physics Conditions</li> </ul>			
😽 Reference Frame	e Specification		
🔤 🤡 Shear Stress Spe	ecification		
😽 Tangential Velo	city Specification		
🕨 🎒 Outlet			
🕨 🐖 Right Wall			
🕨 🐖 Symmetry Plane			
🕨 🐖 Top Wall			
Physics Conditions			
Physics Values			
🗝 🖻 Derived Parts			
🖻 📄 Solvers			
· ~ ~ · · · ·			
Shear Stress Specification - Properti	es ×		
✓ Properties			
Method	No-Slip		•
	No-Slip		
	Slip		
	Specified-Shear		
Method			0

Figure 42: Shear stress specification

5.3. Perform the last two steps for the both the Right Wall and Top Wall boundaries.

#### 5.5 Set the Stopping Criteria

Stopping criteria for this simulation will consist of a maximum number of iterations and a minimum limit for the conservation equation residuals. Note that the criteria that are set below are nominal and work on most current desktop computers. There have been situations where the residual level is not met on some PC hardware. If this is the case when you run the simulation, you may need to adjust the criterion value.

- 1. Expand the Stopping Criteria node and note that the two default criteria (Maximum Steps and Stop File) for steady simulations are present. Automatically generated stopping criteria cannot be deleted, but the Enabled Property can be activated or deactivated.
- 2. Select the Stop File node and uncheck the Enabled property to disable the criterion.

Simulation Scene/Plot		čg  ▼
duct-lam		
<ul> <li>Geometry</li> </ul>		
Continua		
Regions		
Derived Parts		
Solvers		
<ul> <li>Stopping Criteria</li> </ul>		
Maximum Steps		
Solution Histories		
Solution Views		
Benorts		
Monitors		
Plots		
Scenes		
Lavout Views		
Summaries		
Representations		
Tools		
Stop File - Properties $\times$		-
<ul> <li>Properties</li> </ul>		
Enabled		
Stop Inner Iterations		
Path	ABORT	
Logical Rule	Or	-
Criterion Satisfied		
Tags	0	
		-
Enabled		

Figure 43: Stop file disable step

3. Select the Maximum Steps node and change the Maximum Steps property to 6000. Leave the Logical Rule property as Or.

duct-lam ×		-
Simulation Scene/Plot		te i
😽 duct-lam		
Geometry		
Continua		
Regions		
Derived Parts		
Solvers		
Stopping Criteria     Maximum Steps     Stop File		
Solution Histories		
Solution Views		
🗎 🗎 Reports		
Monitors		
Plots		
Scenes		
🗎 🖿 Layout Views		
🖻 🖻 Summaries		
<ul> <li>Representations</li> </ul>		
Tools		
Maximum Steps - Properties $\times$		-
<ul> <li>Properties</li> </ul>		
Enabled		
Maximum Steps	6000	
Logical Rule	Or	•
Criterion Satisfied		
Tags	0	
Maximum Steps		6
Maximum number of steps. Range: [1	)	

Figure 44: Maximum steps setting

4. Create the stopping criterion for the continuity equation residual:4.1. Right-click the Stopping Criteria node and select New Monitor Criterion.



Figure 45: New monitor criterion specification

4.2. In the *Select Monitor* dialog that appears, select the **Continuity** residual. Click **OK** to close the dialog box.

Select Monitor		>
🖓 📜 🔚 🖪 🖕 Filter by Path		₽.* 🔳 🗆
Monitors		
Continuity		
O Physical Time		
○ S-momentum		
<ul> <li>Y-momentum</li> </ul>		
<ul> <li>Z-momentum</li> </ul>		
of 6 colocted		
or o selected		
	ОК	Cancel <u>H</u> elp

Figure 46: Select monitor dialog box: Continuity

4.3. Select the newly created Stopping Criteria > Continuity Criterion and change the Logical Rule property to And.



Figure 47: Continuity criterion logical rule

Simulation Scene/Plot		ීස	ŀ
😽 duct-lam			,
F 🖻 Geometry			
🕨 🚞 Continua			
► 🖻 Regions			
- 🖻 Derived Parts			
► 🖻 Solvers			
🔻 🚞 Stopping Criteria			
👻 👥 Continuity Criterion			
Minimum Limit			
Maximum Steps			
Stop File			
- 🚞 Solution Histories			
Solution Views			
- 🖴 Reports			
Monitors			
Plots			
Scenes			
📄 Layout Views			1
- 🗎 Summaries			
Representations			
Minimum Limit - Properties $\times$			-
✓ Properties			
Minimum Value	1.0E-6		
Minimum Value			0

4.4. Select the Minimum Limit and set the Minimum Value to 1.0E-06.

Figure 48: Continuity criterion minimum limit

5. Create stopping criteria for the X-, Y-, and Z-momentum residuals in a similar fashion, with the Logical Rule set to And and the Minimum Value set to 1.0E-6.

# 6 Running the Simulation

Note that if there are any monitoring values or plots that you wish to observe during the simulation, they should be set up before the simulation is run. For this case, post-processing will be demonstrated after the run.

- 1. In the toolbar, click the  $\triangleright$  (Initialize Solution) or use the Solution  $\rightarrow$  Initialize menu item to manually initialize the solution. If your had existing visualizations, you could use them to examine the initial conditions at this point.
- 2. Run the simulation by clicking  $\stackrel{\checkmark}{\checkmark}$  (Run), pressing <Ctrl><R>, or use the Solution  $\rightarrow$  Run menu item. You may also use the  $\stackrel{\checkmark}{\land}$  (Step) to step through the iterations one by one.
- 3. A residual plot will automatically be created and you may watch it as the solution progresses to convergence.
- 4. After the solution has converged, try clearing it using the Solution → Clear Solution... menu item (keep the Fields box checked to clear solution fields). Then run the solution again, stopping it partway through with the (Stop) button on the toolbar. Press (Run) again and notice that the solution continues iterating from where it was stopped.

# 7 Post-Processing the Simulation in **STAR-CCM+**

If existing simulation surfaces are not sufficient to use in plotting, visualizing, or reporting data, Derived Parts are created. Here, we will create a line probe, which is used to sample the velocity field at cells along the Outlet boundary. A visual for the velocity field along the Symmetry Plane will also be created.

- 1. Create the line probe derived part:
  - 1.1. Open the Geometry Scene 1. We must change the view to properly visualize the line probe we will use this opportunity to find out how to achieve specific views within the *Graphics Window* and save them for later use. Select the  $\bigcirc$  (Save-Restore-Select Views) button in the toolbar, and navigate to Views  $\rightarrow +X + Y + Z \rightarrow Up + Y$ .



Figure 49: Geometry scene view setting

Q	♦ ♦ 🔊 🔶 🖽 🌐	¢	▋◀▐▶●
	Store Current View		
	Restore View	>	
	Projection Mode	>	View 1
	Views	>	
	Standard Views	>	
	View Coordinate System	>	

Figure 50: Restore view menu

1.3. Right-click on the Derived Parts node and select New Part  $\rightarrow$  Probe  $\rightarrow$  Line...

duct-lam ×			-	Geometry Scene 1 ×
Simulation Scene/Pl	ot	0 L	:  ▼	Simcontors
🕞 duct-lam				Sincenter 5
🕨 🚞 Geometry				
🕨 🚞 Continua				
🖻 🖻 Regions				
Derived Parts	New Deat	Cell Conferen		
Solvers	New Part /	Cell Surface		
🕨 🚞 Stopping C	Edit	Isosurface		
📄 Solution Hi	New Group	Probe	>	Point
Solution Vi	Group By	Section	>	Line
Reports	UnGroup	Streamline		Presentation Grid
Monitors	ondroup	Constrained Streamline		Arbitrary
Plots	Paste Ctrl+V	Vortex Core	T	
Scenes	Refresh	Separation/Attachment Line		
Layout Viet		Threshold		
Summaries		(m',theta) Warp		
Representatio	ins	Resampled Volume		
Ioois		Warn	>	
		Average Surface		
		Average SuildUe		
		Average Profile		

Figure 51: New probe line menu

This should open an Edit window automatically. If it does not, with the mesh or geometry scene open in the *Graphics Window*, right-click the Outlet Velocity node and select Edit Part in Current Scene.

1.4. Click the Select... button to select Input Parts for the line probe.

Simulation Scene/Plot Edit	ê  ▼
Create Line Probe	
Input Parts	
[Duct Body]	Select
Snap to Part	

Figure 52: Select button for probe line creation

This action will open the Select Objects dialog box.

1.5. De-select the Duct Body region by clicking twice on the side of the item-tree. Then, select just the Outlet boundary.



Figure 53: Select outlet boundary

1.6. Enter (2.0, 0.0, 0.0) m and (2.0, 0.0, 0.3) m respectively for Point 1 and Point 2. Set the Resolution to 14 – this is the number of points the line probe will sample. Since there are only 14 cells in the *z*-direction, having greater than 14 points is unnecessary. Set the Display to the No Displayer radio button. The line you are describing should be visible in the Graphics Window as a barbell.



Figure 54: Line probe options

- 1.7. Click Create at the bottom of the Edit window, and then click Close.
- 1.8. Rename the newly created Line Probe derived part by right-clicking and selecting Rename, or selecting it and pressing <F2>. Rename the part to 'Outlet Line'.
- 2. Create an XY Plot to visualize the velocity field at the outlet:
  - 2.1. Right-click the Plots node and select New Plot → XY Plot The newly created XY Plot 1 node should open and expand automatically when it is created, and a new window should appear in the *Graphics Window* titled 'XY Plot 1'.
  - 2.2. With the XY Plot 1 node selected, open the Parts property selection by clicking the [...] (Property Customizer) button.

Simulation Scone/Plot	9	
Simulation Scene/Plot	q	5 .
- 🛅 Solution Histories		
Solution Views		
- 🗎 Reports		
Monitors		
Plots		
Plot Colors		
Residuals		
▼ 🖄 XY Plot 1		
X Type		
Y Types		
Data Series		
Axes		
<ul> <li>Legend</li> </ul>		
<ul> <li>Color Assignment</li> </ul>		
Annotations		
F Stephate		
I avout Views		
Bepresentations		
Tools		
XY Plot 1 - Properties ×		-
<ul> <li>Properties</li> </ul>		1
Title	XY Plot	
Parts	0 7.	
Representation	Volume Mesh	•
Tags	0	
<ul> <li>Expert</li> </ul>		
Hover Interactions	$\checkmark$	
Show Highlights		
Title Font	Siemens Sans Global Plain 14	
Footer		
Footer Font	Siemens Sans Global Plain 10	
Data Series Order	0	
	all a	

Figure 55: Part selection for XY plot

The XY Plot 1 - Parts dialog box will open.

2.3. Select the  $\mathsf{Outlet}\ \mathsf{Line}\ \mathrm{part}\ \mathrm{and}\ \mathsf{click}\ \mathsf{Close}\ \mathrm{to}\ \mathrm{exit}\ \mathrm{the}\ \mathrm{dialog}\ \mathrm{box}.$ 

-	
	Duct Body
	miniet
	📅 Left Wall
	II Outlet
	🗰 Right Wall
	🐖 Symmetry Plane
	🗰 Top Wall
	🚧 Outlet Line

Figure 56: Outlet Line selection

2.4. Expand the XY Plot  $1 \rightarrow X$  Type node and select the Vector Quantity node. Change the Value property to [0,0,1] m (this is the direction of the line probe).

ductiam x		_
Simulation Scene/Plot	ť	1
Solution Histories		^
Solution Views		
Reports		
Monitors		
Plots		
Plot Colors		
Residuals		
+ 🗠 XY Plot 1		
🔻 🚞 X Type		
Vector Quant	lity	
Y Types		
Data Series		
Axes		
Legend		
<ul> <li>Color Assignment</li> </ul>	nt	
Annotations		
<ul> <li>A Update</li> </ul>		
Scenes		
🗆 🧰 Layout Views		
- 🚞 Summaries		
Representations		
Vactor Quantity Bronartics		-
Properties		
/alue	[0.0.0.0.1.0] m	
and c	[0.0, 0.0, 1.0] III	++
/alue		0
Vector value		

Figure 57: Vector quantity specification

2.5. Select the Y Types  $\rightarrow$  Y Type 1 node and engage the Smooth Values option. This activates solution interpolation, and is an extremely important step to generally include when using XY Plots.

duct-lam ×			-
Simulation		ដៃ	ŀ
marts mesnes			,
Physics 1			
Regions			
<ul> <li>Derived Parts</li> </ul>			÷
Solvers			1
Stopping Criteria			1
- 🧰 Solution Histories			1
Solution Views			1
- 🖴 Reports			1
Monitors			1
Plots			1
- 💬 Plot Colors			1
Residuals			1
★			1
🕨 🚞 X Type			1
👻 🛅 Y Types			1
🗢 🚞 Y Type 1			1
<ul> <li>Scalar Function</li> </ul>			1
► △ Velocity Profile			1
- 🚞 Data Series			1
🕨 🧰 Axes			1
- • Legend			1
<ul> <li>Color Assignment</li> </ul>			1
- California Annotations			1
► 🕂 Update			1
Scenes			1
🗆 🧰 Layout Views			
- 🖻 Summaries			
Type 1 - Properties ×			
Properties			
-Axis	Left Axis		-
lata Type	Scalar		
mooth Values			
ags	1		Ī,
Expert	u		í
lata Focus	None		
and a state of the last second s			

Figure 58: Select smooth values for plot

2.6. Expand the Y Types  $\rightarrow$  Y Type 1 node and select the Scalar Function node. In the *Field Function* property, click [...] to open the Scalar Function - Field Function dialog box. This provides all of the scalar field functions available to be plotted.

duct-lam ×		
Simulation Scene/Plot		ĉe l
Solution Histories		
Solution Views		
🖻 Reports		
Monitors		
🔻 🚞 Plots		
Plot Colors		
Residuals		
\star 🖄 XY Plot 1		
👻 🚞 X Туре		
Vector Quantity		
🕈 🚞 Y Types		
🕈 📄 Ү Туре 1		
<ul> <li>Scalar Function</li> </ul>		
► <u></u> Outlet Line		
Data Series		
Axes		
<ul> <li>Legend</li> </ul>		
<ul> <li>Color Assignment</li> </ul>		
Annotations		
Update		
scenes		
Scalar Function - Properties $\times$		
<ul> <li>Properties</li> </ul>		
Field Function	<select function=""></select>	
Units		
Tags	[]	

Figure 59: Scalar field function selection for plot

2.7. In the Scalar Function - Field Function dialog box, select Velocity → Lab Reference Frame → Laboratory → i. This selection is made easier by searching for 'Velocity' in the search bar.

🖒 Scal	ar Function - Field Function	×
$\nabla$	🖬 📰 🕒 💼 Velocity	× .*   ©   🗖 🗖
•	Field Functions	
0 % 0 0 % 0	$ \begin{array}{l} -r_{s} \text{ Axial velocity} \\ -r_{s} \text{ Cell Relative Velocity} \\ -r_{s} \text{ Radial Velocity} \\ -r_{s} \text{ Relative Tangential Velocity} \\ +r_{s} \text{ Relative Velocity} \\ -r_{s} \text{ Relative Velocity} \\ -r_{s} \text{ Tangential Velocity} \end{array} $	
0 9 9 0 9 0 0	$ \begin{array}{c} \begin{array}{c} & & \\ & & \\ & & \\ & & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \end{array} \\ \begin{array}{c} & \\ & \\ \end{array} \\ \end{array} \\ \begin{array}{c} & \\ & \\ & \\ \end{array} \\ \end{array} \\ \begin{array}{c} & \\ & \\ \end{array} \\ \end{array} \\ \end{array} \\ \begin{array}{c} & \\ & \\ \end{array} \\ \end{array} \\ \end{array} \\ \begin{array}{c} & \\ & \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \begin{array}{c} & \\ & \\ \end{array} \\ \end{array} \\ \end{array} \\ \end{array} \\ \begin{array}{c} & \\ & \\ \end{array} \\$	
1 of 1	13 selected	OK Cancel <u>H</u> elp

Figure 60: Vector direction specification

2.8. Expand the Y Type  $1 \rightarrow \text{Outlet Line node and select Line Style}$ . Change the Style property from None to Solid. Similarly, select Symbol Style and change the Style property from Empty Circle to None.

Y Types     Y Type 1     Scalar Function     Outlet Line     Symbol Style     Data Series     Axes     Legend     Color Assignment     Annotations			<ul> <li>Y Types</li> <li>Y Type 1</li> <li>Scalar Function</li> <li>Y Outlet Line</li> <li>Line Style</li> <li>Data Series</li> <li>Axes</li> <li>Legend</li> <li>Color Assignment</li> <li>Annotations</li> </ul>		~
Line Style - Properties ×	-	•	Symbol Style - Properties ×		
Properties			✓ Properties		
Style	None		Shape	O Empty Circle	•
Width	None		Color	None	
Color			Size	— Horizontal Line	
Opacity		•	Width	Vertical Line	
		•	Spacing	X Cross	
			1 3	+ Plus	
		•		💥 Star	
				Filled Square	
				Empty Square	
				Filled Circle	
				<ul> <li>Empty Circle</li> </ul>	
				🔺 Filled Triangle	
Style	?			Filled Diamond	
Specify the line style.			Shape Specify the symbol shape.		0

Figure 61: Line style and symbol style specifications

- 2.9. Select the Axes  $\rightarrow$  Bottom Axis node. Change the Minimum property to 0.0 and the Maximum property to 0.3, to reflect the z-extent of the domain.
- 2.10. Expand the Bottom Axis node and select the Title node. Set the Title property to 'Z [m]'.
- 2.11. Select the Axes  $\rightarrow$  Bottom Axis  $\rightarrow$  Major Labels node and change the Spacing to 0.05.
- 2.12. Expand the Axes  $\rightarrow$  Left Axis node. Change the Title property to 'U Velocity [m/s]' in the Title node.
- 2.13. Open the Left Axis  $\rightarrow$  Major Labels node and change the Spacing property to 5.0E-5. Also change the Notation property to scientific.

Axes		
👻 🖨 Bottom Axis		
• Title		
<ul> <li>Major Labels</li> </ul>		
Minor Ticks		
👻 🚞 Left Axis		
Title		
··· 🔍 Major Labels		
<ul> <li>Minor Ticks</li> </ul>		
• Legend		
<ul> <li>Color Assignment</li> </ul>		
Major Labels - Properties ×		
✓ Properties	_	· · · · ·
Visible		
Grid Visible		
dia visible	•	
✓Expert		
✓Expert Font	Siemens Sans Global Plain 12	
Font Precision	Siemens Sans Global Plain 12	
▼Expert Font Precision Auto Precision	Siemens Sans Global Plain 12 2	
▼Expert Font Precision Auto Precision Pad Decimals	Siemens Sans Global Plain 12 2	•••
ekspert Font Precision Auto Precision Pad Decimals Spacing	Siemens Sans Global Plain 12 2 2 5.0E-5	
ekspert Font Precision Auto Precision Pad Decimals Spacing Auto Spacing	Siemens Sans Global Plain 12 2 5.0E-5	•••
<pre>kind visible </pre> Expert  Font  Precision  Auto Precision  Pad Decimals  Spacing  Auto Spacing  Notation	Siemens Sans Global Plain 12 2 5.0E-5 Best Fit	
Value value ✓ Expert Font Precision Auto Precision Pad Decimals Spacing Auto Spacing Notation Grid Style	Siemens Sans Global Plain 12 2 5.0E-5 Best Fit Best Fit	•••
Vectorial de la constante de l	Siemens Sans Global Plain 12 2 5.0E-5 Best Fit Best Fit Decimal	

Figure 62: Major labels specification

The plot should appear as below:



Figure 63: U-velocity profile at the outlet.

2.14. Export the U-velocity profile data as a \*.csv for post-processing in another program (e.g. Gnuplot): right click anywhere in the XY Plot 1 scene, and select Export.... Enter an appropriate name and location in the *Save* dialog.

- 3. Create a vector scene of the velocity field on the symmetry plane:
  - 3.1. Right-click on Scenes and select New Scene  $\rightarrow$  Vector. A vector scene should appear in the *Graphics Window*.
  - 3.2. Expand the newly created Vector Scene  $1 \rightarrow$  Vector 1 node and select Parts using [...].
  - 3.3. In the Parts dialog box, select the Symmetry Plane region and press OK.
  - 3.4. Select the Vector  $1 \rightarrow$  Vector Field node and change the Function property to Velocity.
  - 3.5. Select the Vector  $1 \rightarrow$  Color Bar node and change the Color Map to purple-red basic (large difference) to better visualize the velocity.
  - 3.6. Select the Vector  $1 \rightarrow \text{Glyph} > \text{Relative Length}$  node and change the Glyph Length (%) to 2.5 to better visualize the velocity.

The scene should appear as below:



Figure 64: Velocity field at the symmetry plane.

- 3.7. Save the vector scene as an image: right-click anywhere in the Vector Scene 1, and select Hardcopy... Enter an appropriate name and location in the *Save* dialog that appears. Note that a variety of image file types are available, and the resolution of the image may be adjusted.
- 4. Clear the solution, then initialize and re-run or step through it with Outlet Velocity Profile or Vector Scene 1 open watch the velocity field change as the solution converges.

# 8 Further Exploration

In order to gain more experience using STAR-CCM+, you can complete the following additional tasks.

- 1. Restart the flow calculation and converge to a tighter tolerance.
  - 1.1. Load the duct-lam.sim file.
  - 1.2. Export the outlet velocity profile for post-processing in an external software by right-clicking on XY Plot 1 and selecting Export... and then save as duct-lam-1E-06.csv.
  - 1.3. Open the stopping criteria and change:
    - Maximum iterations to  $500\,$
    - Minimum limit for all residuals to 1.0E-8.
  - 1.4. Create a new dataset by expanding the Tools node and right-clicking on Tables, then selecting New Table...  $\rightarrow$  File Table and open the duct-lam-lE-06.csv file.
  - 1.5. Open the XY Plot 1 node and right-click on Data Series, then select Add Data. In the Add Data Providers to Plot dialog box, select the duct-lam-1E-06 dataset and press OK. Open XY Plot 1 in the Graphics Window to view the results.
- 2. Add the energy equation calculation and thermal boundary conditions.
  - 2.1. Load the duct-lam.sim file.
  - 2.2. Right-click on the Continua  $\rightarrow$  Physics 1 node and press Select Models.... Select the Segregated Fluid Temperature model from the list of Optional Models.
  - 2.3. Open the Continua  $\rightarrow$  Physics  $1 > Models \rightarrow Liquid \rightarrow H20 \rightarrow Material Properties and note that Specific Heat and Thermal Conductivity were added. Leave the default values.$
  - 2.4. Also notice that Static Temperature has been added to the Initial Conditions and that Maximum and Minimum Allowable Temperature have been added to the Reference Values. Leave the default values.
  - 2.5. Expand the lnlet boundary node and then open the Physics Values  $\rightarrow$  Total Temperature node. Verify the Value property is set to 300 K.
  - 2.6. For each of the Left, Top, and Right Wall boundaries, change the Physics Values  $\rightarrow$  Thermal Specification from Adiabatic to Heat Flux. Set the Physics Values  $\rightarrow$  Heat Flux to a constant value of 666.67 [W/m<sup>2</sup>] for each wall.
  - 2.7. Add a new Stopping Criterion for the energy conservation equation as done for the other conservation equations.
  - 2.8. Clear and re-run the simulation.
  - 2.9. Create a contour plot of the temperature field at the outlet face: Create a new Scalar Scene. Set the Parts for both the Outline 1 and Scalar 1 displayers to [Duct Body: Outlet]. Set the Scalar Field to the Temperature field function.
  - 2.10. Create a new plot of the temperature profile at the outlet using the existing line probe (created for the velocity profile).
  - 2.11. Create a new line probe that goes down the center of the duct, and then create a chart that plots temperature along this line.