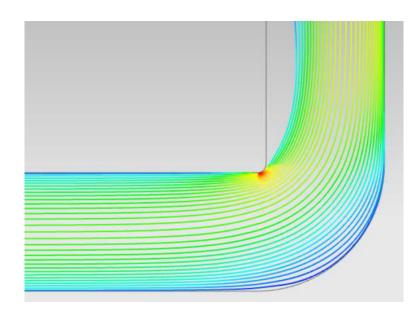
February 2021



How to set-up and run a 2D Flow Simulation in Siemens Simcenter NX12



François Rigo - Dimitri Arendt Faculty of Applied Sciences

Table des matières

1	Create the fluid domain 1.1 Create an empty model	5 5
	 1.2 Create a 2D sketch	6 7 9 10 10
2	Mesh the body2.1Create a mesh file2.2Verify that the different split bodies are connected2.3Add the meshing constraints on the body edges2.4Create 2D meshes2.5Create 2D dependent meshes on the other face2.6Create 3D swept mesh	13 13 14 14 16 17 18
3	Specify material properties and set the simulation constraints3.1Create a sim file	19 19 19 19 21 21
4	Solve the simulation4.1Set the solution attributes4.2Set the solver parameters4.3Solve the simulation	23 23 23 24
5	Analyze the simulation results5.1Review the verbose5.2Check the created files5.3Set-up key measurements for rapid analysis5.4Plot the results	25 25 25 25 26
6	Change the design	30
7	Change the turbulence model or solution type 7.1 Turbulence model 7.2 Solution type	32 32 32
8	Troubleshooting	33
9	Extra Resources	34

Summary and work-flow

This tutorial explains the work-flow to set-up and run a 2D Flow Simulation in Simcenter NX12. Its targeted audience is anyone without any particular prior knowledge of NX Simcenter Environment. It explains in details the basic actions in order to perform a simple fluid simulation. It is meant to be used as a quick-guide to lead rapidly to a first solution. The reader can off course refer to the official NX help for any required further explanation or when his needs deviate from the simple example presented in this document. The tutorial is based on a simple test case : the study of a 2D turbulent incompressible flow in curved pipe. The following list explains in brief the work-flow and main steps needed to perform a flow simulation in SimCenter NX12.

1. Create the fluid domain (.prt file)

- The fluid domain for the simulation has to be created by using the internal CAD within Simcenter NX. Basically, for a 2D simulation, the geometry is completely defined with the sketcher. A solid body is then created by extruding the sketch to a small thickness.
- To prepare the meshing, the solid body can be split into multiple simple geometric shapes.
- $-\,$ At the end of this step, you shall have a saved <code>yourproject.prt</code> file.

2. Mesh the body (. fem file)

- The fluid domain is meshed using the internal mesher of Simcenter NX.
- First, meshing constraints have to be set on the edges of the bottom face of the body : number on edge, size on edge or biasing on edge (for a boundary layer).
- Then, 2D mapped meshes have be created on all faces of the bottom face of the body.
- The 2D meshes are then copied onto the other face and swept along the thickness of the part in order to generate 3D meshes with only one element along the thickness.
- At the end of this step, you shall have a saved yourproject_fem.fem file

3. Specify material properties and set the simulation constraints (. \mathtt{sim} file)

- Different simulation objects have to be created : fluid materials, boundary conditions, initial conditions. At this stage, some measurements of interest for the simulation analysis have to be defined : force on a face, min/max velocity/pressure within a given area,...
- At the end of this step, you shall have a save yourproject_sim.sim file, ready to be "solved".

4. Solve the simulation

- The solution attributes have to be set : steady state or transient, turbulence model, use wall function or not, data fields to be retrieved,...
- The solver parameters have to be set : relaxation time step, convergence criteria, number of iteration limits,...
- Finally the simulation can be run. At the end of this step, you shall have run the simulation and the solution shall have converged (or not) to a result after some iterations.

5. Export and analyse the simulation results

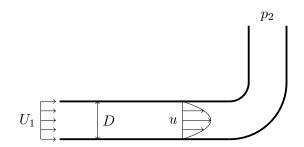
- The solution verbose and the convergence graph should be reviewed within the "Solution Monitor" window.
- The results can be loaded in order to plot the field of interest : velocity, pressure maps, 2D graph along a path, streamlines, ... A html report with extra results (force on a face for instance) is also generated if it was defined at step 3.
- At the end of this step, you can save again your working .sim file as a reference before experimenting with different meshes, constraints or solver parameters.

Each step will be explained with more details using a practical example in the following chapters. Extra documentation about workflow for Flow simulation can be found here :

https://docs.plm.automation.siemens.com/tdoc/nx/12/nx_help#uid:xid1128419:index_advanced:id1245911: id629501

Physical context of the tutorial

We consider a 2D flow in a pipe (unit depth) of water (incompressible, viscous, steady-state). The velocity at inlet is uniform at $U_1 = 0.34$ m/s and the outlet is at ambient pressure ($p_2 = 0$ Pa in relative). The pipe diameter is D = 65 mm and its total length is L = 2.78 m. This tutorial will compute the entire flow but under some assumptions, we can estimate theoretically the pressure drop $\Delta p = p_1 - p_2$ between the inlet and the outlet, due to viscous loss.



Because the fluid is viscous, it is not allowed to use Bernoulli equation. Nevertheless, by neglecting entrance effects and taking the mean velocity $\bar{u} = U_1$, the pressure loss can be computed with the Darcy–Weisbach equation (adapted from a 3D cylindrical pipe to a 2D plane pipe),

$$\Delta p = f \frac{L}{4D} \rho U_1^2$$

with f the Darcy friction factor, obtained (experimentally) for a turbulent flow in a smooth pipe (here, $Re=\frac{U_1D}{\nu}=2.2\ 10^4>2000$ thus turbulent) with $\frac{1}{\sqrt{f}}=2\log(Re\sqrt{f})-0.8$ or with Moody diagram. We found f=0.0253 and thus $\Delta p=31.2$ Pa.

Physically, the most important part to understand in the numerical setup in the boundary conditions choice. The fluid is entirely described by Navier-Stokes (NS) equations (differential equations for 4 unknowns : 3 components for \underline{u} and 1 for p). 4 boundary conditions are thus needed to solve the computation. Velocity and pressure are linked together thanks to NS equations, when one is known, the other to. Boundary conditions are needed for the whole control surface. Here, the velocity is imposed at the inlet (3 values) and the pressure at the outlet (1 value). The rest of the control surface is a wall, for which the condition is a *no slip wall* (here, we have a viscous flow, with zero velocity at the wall). When the flow is turbulent, a special treatment of the wall has to be done, using a *wall function* (modelling the turbulent boundary layer, more details in section 7.1. If the flow is inviscid the condition is a *slip wall* (or equivalently, impose a zero velocity across the wall).

1 Create the fluid domain

Before to open NX12, be sure that you are connected to the network of ULiege (via the VPN BIG-IP Edge CLient). If you connect for the first time, type *Licensing Tool* in Start Menu and select these bundles. Afterwards, you can start NX12.

Siemens Available Bundles: Appled Bundles: Borrowing Settings Fibersim for NX Academic Fee NX Academic Renewal Fee Fibersim for V5 Academic Fee NX Academic Bundle 1yr CAE+ About Licensing Tool Bundle Components: Siemens PLM Software Licensing Tool — — Siemens PLM Software Licensing Tool — — Bundle Settings Ellicense Server Eldit Borrowing Settings Ellicense Server Edit Somowing Settings Ellicense Server Edit About Licensing Tool Ellicense Server Edit Semens Settings Ellicense Server Edit Semers Licensing Tool Successful Successful Susuer: Successful Susuer: Successful Susuer: Successful Susuer: Successful Susuer: Successful Susuer: Successful Susuer: Successful Susuer: Susuer: Webkey: GQSMYSM368 Issuer: Superson: Semens Licensing Version: 11.13 Semens Licensing Version: 8.0	🔊 Siemens PLM Software Licensing	Tool	- 🗆 X
Rundle Settings Fibersim for NX Academic Fee NX Academic Renewal Fee Image: Settings Fibersim for V5 Academic Fee NX Academic Renewal Fee Image: Settings Image: Settings NX Academic Renewal Fee Image: Settings Image: Settings Image: Settings Image: Settings Image: Settings <td< th=""><th></th><th></th><th>SIEMENS</th></td<>			SIEMENS
Image: Service server Bundle Settings Edit Image: Service server License Server 28000@pegase.ltas.ulg.ac.be Edit Image: Service server S000@pegase.ltas.ulg.ac.be Edit Image: Service server S000@pegase.ltas.ulg.ac.be Edit Image: Service server Sold To ID: 1471462 - Universite de Liege Image: Service server Sold To ID: 1471462 - Universite de Liege Image: Webkey: GQSMYSM368 Issuer: SIEMENIS Version: 11.13	 Borrowing Settings Environment Settings About Licensing Tool 	Fibersim for NX Academic F	Fee NX Academic Renewal Fee NX Academic Bundle 1yr CAE+ Bundle Components:
Bundle Settings License Server: 28000@pegase.ltas.ulg.ac.be Edit Borrowing Settings 28000@pegase.ltas.ulg.ac.be Edit Environment Settings Sold To ID: 1471462 - Universite de Liege Webkey: GQSMYSM368 Issuer: SIEMENS Version: 11.13			SIEMENS
	Borrowing Settings	License Server: 28000@pegase.ltas.ulg Connection status: Sold To ID: Webkey: Issuer: Version:	a.ac.be Successful 1471462 - Universite de Liege GQ5MYSM368 SIEMENS 11.13

1.1 Create an empty model

File > New > under tab "Model" select "Model" > OK

Be sure to create the file inside a new folder, on your Desktop for example, or in a place where you have written access. All next files will be linked to this first file. when NX will computes results, they will be saved in this chosen folder.

NX	🗋 🤌 🗋 🔹 🛷 📅 Switch Wir	New						ა x	EMENS _ 🗗 🗙
File	Home Tools	Machining Line Planner	Manufacturing	Inspection	Mechatronics Conce	ept Designer Pr	ress Line Line Designer	Ship Structures	nmand 🔎 📃 🐟 🕜
) 🤌 🙈 🎜 📗	Ship General Arrangement	Model	DMU	Drawing	Layout	Simulation Additiv	e Manufacturing	
New Open Open a Assembly Templates A Preview A									
	Standard Filters								
_	lenu 🕶				Units Millin	meters 👻			
¢	Reuse Library	Name	Type	Units	Relationship	Owner	Λz		
	Name	Model	Modeling		rs Stand-alone	AUTORITE	1		
"	Reuse Examples	Assembly	Assembli		rs Stand-alone	AUTORITE	Y		
	- Tastener Assembly Configuration	Shape Studio	Shape Stu		rs Stand-alone	AUTORITE	Liter I		
~	🗄 👷 Favorites	Sheet Metal	Sheet Me		rs Stand-alone	AUTORITE			
1		Routing Logical	Routing L		rs Stand-alone	AUTORITE	AX		
		Routing Mechanical		Aecha Millimete		AUTORITE			Around
0		Routing Electrical		lectrical Millimete		AUTORITE	Properties	~	
		Blank	Gateway		rs Stand-alone	none	Name: Model		
							Type: Modeling		
(Units: Millimeters		
							Last Modified: 09/11/2017	06:07 AM	
3							Description: NX Example w		
							Description. W/ Example w		
		New File Name						^	
		Name model1.prt							
	Folder Home\Desktop\tuto\								
		Part to reference						^	
		Name							
									and the second sec
	<								1 11 11
Search OK Cancel									
	Member Select								
Preview V									
_									

NX 🖬 🔊 - 🍽 🖗 🛍 🗋 🗆 - 🛷 🎛 Switch	Window 🗌 Window 🕶 🗟	NX 12 - Modeling	SIEMENS _ 🗗 🗙
File Home Assemblies Curve Analysis	View Render Tools Application		Find a Command 🔎 🔳 \land 😗
	Pattern Feature Unite + Shell Edge Feature	More Surface Work on Add Ssembly Add	
Menu ▼ No Selection Filter ▼ Entire Assembly ▼	🖞 % 🗄 • % % 🍕 🛄 • 🔕 😂 🐼 🖊 🗡 🧎 🗛 📈	+ ⊙ ♀ + 🗸 ኛ 🏶 🖩 🖾 🖌 🖋 🖫	🛛 T 🖑 T 🧊 T 🚧 T 👘 T
Part Navigator	💪 model1.prt 🗈 🗙		
Name ▲ Up to Date + ⊕ Model Views + ♥ ♥ Cameras H □ ⊕ Model History			
Model History			
Arborescence (design history)			
		Z	
ð			
0			
Θ			
Ex.			
₿ ₹			
	Z		
Dependencies V Details V			
Preview V	^		
			W (E)

You shall have this view and you are ready to create your part.

NX12 is based on an tree structure (the left blank panel) where you can check what you have done, go back and modify previous steps. If you have several files opened in NX12, click on the top tab *Switch window* ^{The Switch Window} to navigate.

To **navigate in space**, you can use rather navigation icons 🗖 🔍 or using a mouse :

- Useful shortcuts are available when holding down the right mouse button.
- Clicking the central mouse button and dragging will rotate the view.
- Clicking the central mouse button, then the right mouse button and dragging will move the view. You can do the same more easily by holding down the SHIFT key and clicking only the central mouse button and dragging.
- Rolling the mouse wheel will zoom in/out.
- As you work in a sketch plane, if you unintentionally moved the view out of plane, you can use shortcut SHIFT+ F8 to get back to the top view. You can also choose a point of view (on a plane, isomeric...) using



1.2 Create a 2D sketch

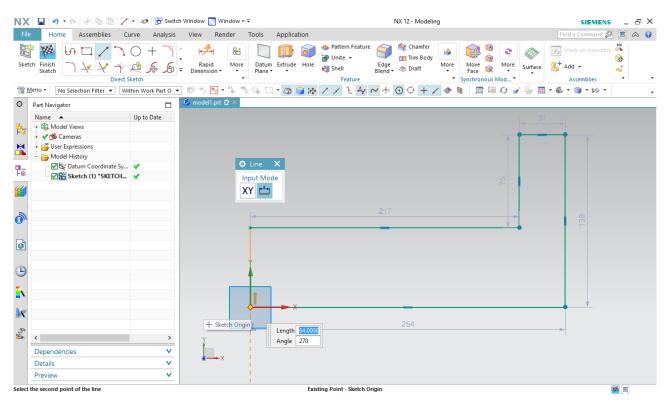
Click "Sketch" in the upper left corner $\mathbb{S}_{\text{steth}}$. You can keep the default coordinate axis > OK

Create Sketch		ਹ X
Sketch Type		^
🛐 On Plane		•
Sketch CSYS		^
Plane Method	Inferred	-
Reference	Horizontal	-
Origin Method	Specify Point	-
Specify CSYS	<u>.</u>	R -
	< 0K > 0	ancel
		uncer

1.3 Draw the fluid domain

Draw the fluid domain and eventual obstacles using the tools under Direct Skecth : Rectangle, Line, Arc,... Here, we have an internal flow, so we draw the fluid inside the pipe. For an external flow around a body, your fluid domain will be what is around the body.

Draw the following sketch using the *Line* \checkmark command. The dimensions do not have to be the same at this moment, only the general shape matters.



Closed sketch : To be sure that your sketch is a **closed contour** (to be able to create a solid body in section 1.5), be sure that you select the same point when drawing intersecting lines (orange point). NX12 helps you to draw vertical or horizontal line by adding some orange dashed lines.

Delete : If you want to delete/modify an element, be sure to deselect the tool you used first (if you not, you will still be in drawing mode). Right click on the element you want to delete and select \times (or click on the element and CTRL+D).

Element selection : The different elements are selected just by clicking them successively, there is not need to hold down CTRL key. To deselect all : press ESC. To deselect one item only : click it with left mouse holding down SHIFT

Dimensions and constraints : You will see that grey dimensions appear automatically in such a way to fully define your sketch. These are auto-dimensions but not constraints so your sketch is free to move. To add constraints, you can either

1. Double-click on grey dimension and adjust the value to make them constraints, they will appear in dark blue color.

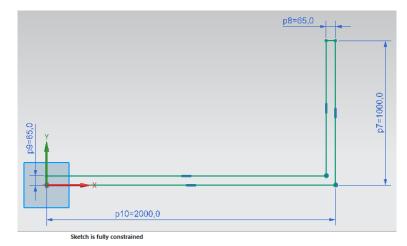
2. Select a segment, right click and then choose horizontal/vertical dimensions



 Add more complex constraints, like parallelism, same length, perpendicularity, etc.. by selecting successively different elements and then choosing suggested constraints. Dimensions constraints can be found under *Direct Sketch* > "Rapid Dimension". You can also find all constraints under *Direct Sketch* > More > "Sketch Constraints"

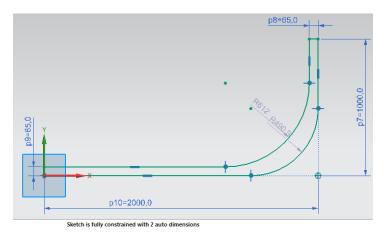
Rapid Dimension •	More	Datum Plane	Here we have	Datum Plane •	Extrude	O Hole	🚸 Pattern Feature 🏠 Unite → 🍕 Shell	Edge Blend •	💱 Chamfer 础 Trim Body 役 Draft	
 ▶⁴ Rapid Dimension D ▶⁴ Linear Dimension ∧^a Radial Dimension ∠1 Angular Dimension 				tly Used metric Co	nstraints		▶ ″⊥ Display	Sketch C	Constraints	
			<i>⊪</i> ⊥ Geo	Constrai metric Co meter Dim	nstraints		[법] Make S > 쒸 Display	-		
				🚰 Display Sketch Auto Dimensions			-			

For this tutorial you should have the following constraints.

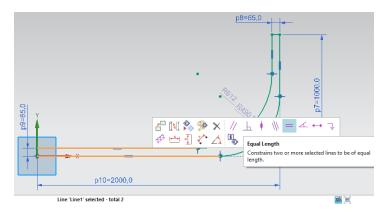


It shouldn't contain any grey dimensions (auto dimensions) but only dark blue constraints (px = ...). This means the sketch is fully constrained. "Sketch is fully constrained" will appear at the bottom of the window. If you have red/purple constraints, it means that your sketch is over-constrained, you should remove some constraints in this case.

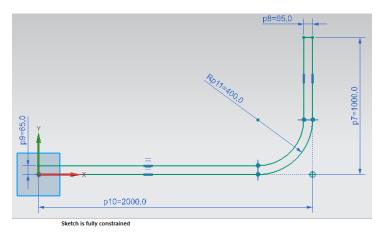
Draw a round corner : Select the *Filet* tool in *Direct Sketch* and then successively the two inner legs of the corner. Repeat with the two outer legs. You should have the following sketch with two additional degree of freedom, being the two arc radius.



To keep a constant section for the pipe, you will select the two horizontal legs of the pipe and force them to the same length as shown-here below. Because of the other constraints, imposing only the two horizontal legs is enough.



The only remaining degree of freedom is the radius of the outer arc that you should force to 400mm. The final sketch should appear as follows.



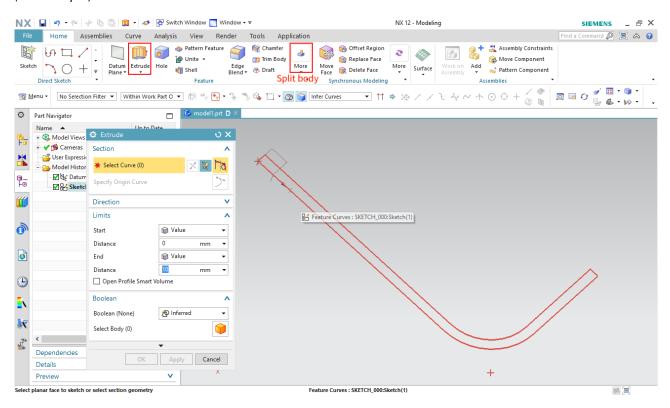
1.4 Finish the sketch

When you are done with the sketch, Click "Finish Sketch" in the upper left corner

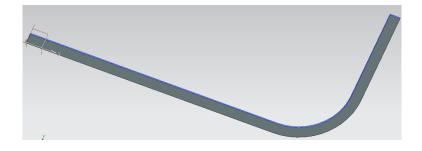


1.5 Extrude

Select your sketch > Click "Extrude" under Feature toolbox > Choose an extrusion length (end distance) of 10mm (for example) > OK



By rotating the view with the middle mouse button you can verify that you have now a 3D solid body (if you did not have close your sketch, it will be a shell).



1.6 Split body to prepare the meshing

To make a consistent meshing with quadrilateral elements, and be able to apply specific mesh constraints on domains, the fluid domain has to be divided in simple geometries : triangles or quadrilaterals, possibly with curved edges. Thus, we will split here the body in 3 parts thanks to two small edges at the connection between the straight entry and the rounded turn.

Under Feature toolbox > More > Trim > "Split body"



Select your body, choose "*Extrude*" under *Tool Option* and then under *Tool* > *Section* > *Select Curve* > *Click on* "*Sketch Section*".

🌣 Split Body	૨ ૪		Create Sketch	ບ :
Target	^		Sketch Type	
 Select Body (1) 	9		ম On Plane	-
Tool	▲ Extrude ▼		Sketch CSYS	
Tool Option Section			Plane Method	Inferred 🔻
✓ Select Curve (2)	🛙 Ha		Reference	Horizontal 🔻
Direction	^		Origin Method	Specify Point 🔻
 Specify Vector 	🛃 🕇 Sketch Sect	ion	Specify CSYS	× /2 -
	OK Cancel			OK Cancel

Draw split limit : After clicking on "*Sketch Section*", you will enter in a temporary sketch, where you have to first select the system coordinate (same as before, CSYS).

You should split the body in three parts by adding two perpendicular lines at the entry and exit of the corner. You should also have this new sketch fully constrained. When you are done with the sketch, Click *"Finish Sketch"* in the upper left corner.

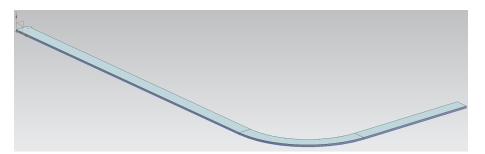
NX = S ? • ? + S S & Y • S Switch Window S Window	iow • •	NX 12 - Sketch		SIEMENS _ 🗆 🗙
Task Home Analysis View Tools				Find a Command 🛛 🖉 🗷 🗛 😝
	O + + Studio Spline ○ Polygon rc Circle Point ○ Ellipse > >Conic ● Offset Curve ▲ Pattern Curve Curve Curve	✓ Quick Quick Chamfer	Kapid Geometric Make Display Sketch Dimension • Constraints Symmetric Constraints • Constraints •	-
™Menu▼ No Selection Filter ▼ Within Work Part Only	▼ ◎ ∿ 號▼ ኈ □ ▼ @ @ @ / / ℃ Ą ~ ↑ @ 0 + / ◈ 8	1 □ □ 0 ≠ 1 = 0 × 0 × 0 × 10 ×		-
Part Navigator	+ tutorial2017.prt ≋ ×			
Name Up to Comm Name Up to Comm Name User Expressi Nodel History Model History Bill Statch (1) ✓ Bill Statch (2) ✓ Bill St	O Line × Input Mode xr ≝		*	Length:/Line13 of SKETCH_000 Angle 0
Dependencies V				
Details V				
Preview V				+
Select the second point of the line		End Point - Line13 of SKETCH_000		第三

Make sure to tick the box for *"Keep Imprinted Edges"* under *Settings* and click OK. By doing so, the software automatically creates Glue Coincident type mesh mating conditions between the bodies when you switch to the FEM file. You should inspect these mating conditions and ensure that they were created at all appropriate locations

Split Body	ວ ×
Target	^
✓ Select Body (1)	9
ТооІ	^
Tool Option	Extrude 🔻
Section	^
✓ Select Curve (2)	
Direction	^
 Specify Vector 	<u>u</u> † ₄ -
	•
	More: Click to see all dialog options.

🌣 Split Body	ى ە	×
Target	/	^
✓ Select Body (1)	9	۲
ТооІ	/	۸
Tool Option	Extrude 👻	•
Section	^	^
✓ Select Curve (2)	11 A	Ø
Direction	^	^
 Specify Vector 	et 💷 📼	•
Settings	/	۸
Tolerance	0.0100	0 C
Keep Imprinted Edges		
Preview	N 1	V
•	OK Cancel	el

When drawing split limits, your elements should referring to the first sketch geometry. By doing so, if you change a dimension in the first sketch later, this split body and the meshing will follow and adapt automatically. If you want to split the body in more complex parts inside a body part, start from the new split body part and re-do a split body procedure (example in section 6. You should end up with this design,



Make sure you save your model as a yourproject.prt file. Do not hesitate to save it often using CTRL+S.

Sketch tutorials can be found here : https://docs.plm.automation.siemens.com/tdoc/nx/12/nx_help#uid:id1251042 https://docs.plm.automation.siemens.com/tdoc/nx/12/nx_help#uid:xid1128417:index_sketcher:id188016:id771117

Sketch video examples can be found here : https://docs.plm.automation.siemens.com/tdoc/nx/12/nx_help#uid:xid1128417:index_sketcher:id1389302

2 Mesh the body

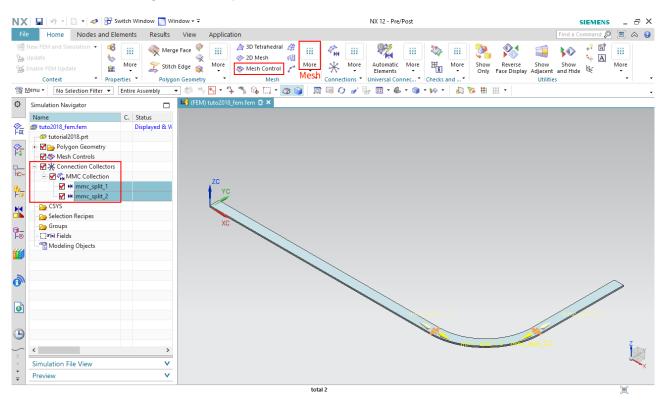
2.1 Create a mesh file

- File > New > under tab "Simulation" select "Simcenter Thermal/Fow" with type "Fem" > OK Be sure to save it in the same folder as the .prt file, under another name (using name_fem.fem for example).
- Keep the box "Associate to Master Part" ticked.
- In the field "Part", you shall select the part you just created.
- Untick the box *"Create Idealized Part"*. This option is used when working in team on the same part or if the part has to be simplified for the simulation but the original design has to kept unaltered.
- Select Solver "Simcenter Thermal/Flow" and Analysis Type "Flow". Click OK.

🗘 New FEM		υx				
FEM Name		^				
tuto2018_fem.fer	n					
CAD Part		^				
Associate to M	Master Part					
Part tutorial201	в 🔻 📔					
Idealized Part		^				
Create Ideal	ized Part					
Bodies		^				
Bodies to Use	All Visible	•				
Polygon Body F	Resolution Standard	•				
Geometry		^				
	Geometry Options					
Solver Environr	nent	^				
Solver	Simcenter Thermal/Flow	•				
Analysis Type	Flow	•				
Mesh Morphin	g	^				
Save Full Morphing Data						
Description		^				
	ок с	ancel				

2.2 Verify that the different split bodies are connected

First thing to do, is to verify that the different polygons created by the split body command are correctly connected to each other. It is OK if there are two elements under *Connection Collectors > MMC Collection > mmc_split* (Mesh Mating Conditions, just check if they are present).



If it appears that two polygon bodies have common face but are not connected, you can add the connection manually by clicking Mesh Mating under Connections toolbox.

NX 🖬 🤊 🕶 🛸 🥙 🖗 Switch W	indow 🗖 Window 🔻 🔻			1	NX 12 - P	Pre/Post							SI	EMENS	×
File Home Nodes and	Elements Results Vi	ew Application											Find a Co	mmand	<i>₽</i> ≈
	Physical Properties More Mesh Collector	Face Edge Te Polygon Geometry ▼	Image: Application of the second	More • * 10	Connec Co k to the total	More Mesh Mating Connects two odies and th	o separate neir associ	solid	"n •	Now Show Only F	Reverse Face Displa	Show Show y Adjacent Utilities	Show	+ ² B More	
Simulation Navigator		🕫 (FEM) tutorial2017_fem.	fem ¤×		21	D or 3D mes		🌣 Me	sh Matino	a Cono	ditions	ა ×			
R Name ● tutorial2017.fem.fem. ● tutorial2017.pt + #≫ Polygon Geometry + #≫ Kosh Controls + #≫ Connection Collectors ● Selection Recipes ● Groups → Fields ● >Modeling Objects	C Status Filter Display (Filter (Filter (Filter (Filter (Filter (Filter (Filter) (Filter) (Filter)							Select Select Param Mesh I Face Se	matic Crea tion t faces or b neters Mating Type earch Optio Distance	odies (2	2) All Pairs 0.0254				

2.3 Add the meshing constraints on the body edges

Mesh Control > choose in the dropbox the constraint you want > Select all edges for which you want to apply the constraints > Click the preview button > OK.

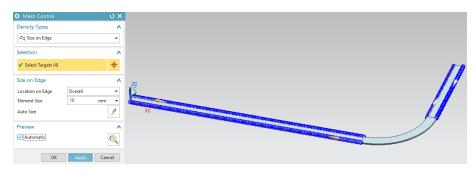
🖄 🗇 🍫	Mesh Control	૨ ૪
3D 2D Mesh Tetrahedral Mesh Control	Density Types	^
Mesh	Number on Edge	•
	Number on Edge	
▼ <a>Image: The second second	Size on Edge	
m.fem ¤ ×	☆Chordal Tolerance on Edge	
	🛠 Biasing on Edge	

Here, we will set the mesh constraints, on one face only (upper face of the extrude, be sure to select always edges from the same face).

Biasing on edge : we will apply on the 4 perpendicular section a "bias on edge" with option "Center of edge", 30 elements and bias ratio of 1.12 to have smaller elements close to the wall where the velocity gradients are more important. The orientation of the orange arrow does not matter here because the biasing is centered. Remark : A matching edge between two polygons should be specified once only. NX12 automatically applies the mesh constraint on both sides.

Mesh Control		υx
Density Types		^
Biasing on Edge		-
Selection		^
🗸 Select Targets (4)		¢
Biasing on Edge		^
Bias Origin	Center of Edge	•
Number of Elements		30 🗘
Bias Ratio	1.12	•
Auto Size		1
Preview		^
Automatic		0
		~
ОК	Apply	Cancel

Size on edge : Next, you force the elements size to be 10mm on the entry and exit lateral sections. You shall put all those constraints in the same plane, here the upper plane.



Repeat the same operation and set a size on edge of 5mm on the outer arc of the corner.

Ø Mesh Control		ა x
Density Types		^
😂 Size on Edge		•
Selection		^
 Select Targets (1) 		ф
Size on Edge		^
Location on Edge	Overall	•
Element Size	5	mm 👻
Auto Size		۶
Preview		^
Automatic		Q
ОК	Apply	Cancel

Number on edge : you shall also set the short edges along the thickness (z-axis) to have one single element, as the simulation is 2D. Do not hesitate to zoom a lot on the thickness and to rotate the view to be sure only the small thickness edge is selected.

Mesh Control	ა x	
Density Types	^	
🔩 Number on Edge	•	
Selection	^	
< Select Targets (8)	÷	ζζ.
Number on Edge	^	
Number of Elements	1 🗘	XC
Auto Size	1	
Minimum Element Size		
Preview	^	
Automatic	<u>@</u>	· · · · · · · · · · · · · · · · · · ·
ОК	Apply Cancel	1

Tips : You can set a constraint on multiple edges at a time. For the biasing on edge (especially useful for boundary layer), you have to select the orientation (start of edge, end of edge or middle of edge) and it varies from edges to edges depending on the edge natural direction.

You can always modify constraints already defined by double click on it in the arborescence. You can also rename the constraint to identify it more easily.

☞ tutorial2017_fem.fem	Mesh Control	ა x
tutorial2017.prt	Density Types	^ ^
+ ☑≥ Polygon Geometry – ☑◆ Mesh Controls	✤ Biasing on Edge	•
Edge Density	Selection	^
- I Bias_on_egde_BL	✓ Select Targets (4)	
 ✓ Lateral_straight ✓ Lateral_corner 	Biasing on Edge	^
☑ Thickness	Bias Origin	Center of Edg∈ ▼
+□*Connection Collectors	Number of Elements	30 🌲
► CSYS	Bias Ratio	1.12 💌 🗸
Selection Recipes	ОК	Apply Cancel
Groups		Apply culleer

For more complex geometry, you can set the view to static wireframe to view all your constraints at once. An edge density is shown with a yellow diamond, see here-under.



2.4 Create 2D meshes

You can now mesh the face by using 2D mapped mesh. Of course you should mesh the plane on which you defined the mesh constraints. Make sure to create one 2D mesh per face at one time (we have 3 bodies in this case), to obtain 3 meshes and be able to use the 2D dependent option (see section 2.5).

Mesh > *More* > "2D *Mapped*" > *Click the face where you set the edge constraints*. Select either *QUAD4* or *TRI3* elements. In this tutorial we select quad (structured) elements. When working with quadrilaterals mesh, if you have correctly set the edge constraints, the "*Element Size*" parameter should have no effect and can be left to a default value. Click the icon next to "*Show Result*" to get the preview before clicking OK.

🕻 🖬 🤊 🕶 💷 🕶 🖉 Switch V					NX 12 - Pre/Post	SIEMENS _
le Home Nodes and	Elements Results	s Vie	w Application			Find a Command 🖉 🛎
le Home Nodes and lew FEM and Simulation • Jpdate nable FEM Update Context • Aenu • No Selection Filter Simulation Navigator Name • tutorial2017.pt • ap-Polygon Geometry • Mesh Controls • Selection Recipes • Groups • Groups • Fields • Modeling Objects	 Manage Materials Physical Properties Mesh Collector Properties Within Work Pa C Status F Display (F (F (F (F (F (F (F 	more Tt Only	Constraints of Carlot Constraints Con	Image: Second		Find a Command P *
< Simulation File View Preview		> *	Preview and Modify Mesh Constr. Show Result Define Corners Edge Density Mesh Options OK Apply Ca	ncel	Ship Mesh Automation Refine Mesh from Field	

Mesh all the other polygon faces of the plane similarly (3 in total). In more complex geometry, make sure not to over-constrain the mesh. At the end, you obtain 3 shells below "2D Collectors". Each shell contains a 2D mapped mesh (3 for the 3 parts of the upper face only) : do not mesh the face along the thickness, this will be done in the 3D mesh section.

	Window 🗖 Window 👻 👻						NX 12 - Pre/P	ost							s	IEMENS	>
File Home Nodes and	d Elements Results	Viev	w Application												Find a O	ommand	/ I ~
% Enable FEM Update Context ▼	 Physical Properties M Mesh Collector Properties 	ore I	Face Edge Polygon Geometry ▼	Tetrahedra	2D Mesh I Mesh Control 2 Mesh	- • •	* 1D Connection Connections	•	Automatic Mo Elements V Universal Con	re 👘	•	🎇 Show Only	Reverse Face Display	Show Show y Adjacent Utilities		+ ⁷ ⊠ ¹ + ⊠ Mc ¹ k	
■ <u>Menu</u> No Selection Filter			▼ 0 % 5 ▼ 3 % 0		ao 🖌 🖗 🖬 🕶 🕹 🕶	r 🎯 🕈 🚧	▼ # % ₩ # *										
Simulation Navigator			(FEM) tutorial2017_fe	m.fem ¤×													
A 1	C CL																
🏠 Name	C Status Filt	er 🚬															
% Name ■ #tutorial2017_fem.fem	Display	er															
-		er															
≠ #tutorial2017_fem.fem	Display																
■ tutorial2017_fem.fem = tutorial2017.prt	Display Not Lo	ter														٨	
	Display Not Lo (Fil	ter ter														A	
■ tutorial2017_fem.fem □ tutorial2017.prt □ tutorial2017.prt □ ≫ Polygon Geometry □ ≫ 2D Collectors	Display Not Lo (Fil (Fil (Fil	ter ter							4							A	
■ tutorial2017_fem.fem □ tutorial2017_prt □ tutorial2017_prt □ polygon Geometry □ ∞ 2D Collectors □ ∞ Shell(1)	Display Not Lo (Fil (Fil (Fil	ter ter ter							A							4	A
tutorial2017_fem.fem tutorial2017_prt two-rial2017_prt we > Polygon Geometry we > 2D Collectors we Shell(1) we 2d_mapped_mesh	Not Lo (Fil (Fil (Fil (Fil	ter ter ter						<i>f</i>								4	A
tutorial2017_fem.fem tutorial2017.ptt +∞aPolygon Geometry -∞e>2D Collectors -∞sShell(1) +∞sShell(2)	Display Not Lo (Fil (Fil h(1)) Eleme (Fil	ter ter ter															Ą

Every time you generate a 2D mapped mesh with quadrilateral elements, the constraint of one edge is reported to the opposed edge. This constraint appears as "Mapped Mesh Edge Density" in the arborescence and as blue diamond in the main view (see image here- above). This also means there might be a logical order on how to mesh the different faces.

2.5 Create 2D dependent meshes on the other face

In order to have a pure bidimensional problem, the mesh has to be identical on both upper and lower faces of the solid. To save time, we can copy paste the upper 2D mesh of each upper face on the bottom one, thanks to 2D dependent mesh option (proceed for the 3 existing 2D meshes separately).

Mesh > *More* > "2D *dependent*" > *Select the master (upper) face (already meshed)* > *Select the corresponding face on the opposite side (lower) as the target face* > *OK.*

Repeat the process for all 2D meshes separately. Sometimes, the selection of the upper face as the "Master Face" is not possible/easy : in this case, check that you have created one 2D mesh for each upper face (see section 2.4) and rotate the view in 3D to be able to select it.

🗶 🖬 🤊 👻 🎯 👻 🥔 😵 Switch V	Vindow 🗖 Window 🕶	-					NX 12 - Pre/Po	st							s	IEMENS	- 🗆
ile Home Nodes and	l Elements Resu	ilts Vie	ew Application												Find a Co	ommand	P
	-		Image: Stitch Image Image: Stitch Image:		 2D Mes Mesh Cont 		More * 1D Connection		₩ Automatic Elements	More	More	Now Show Only F	Reverse ace Display	Show Adjacent		+ @ Mo + @ Mo	
Context -	Properties		Polygon Geometry 👻		Mesh		Recently Used				•			Utilities			•
Menu - Polygon Face	 Within Work 	Part Only	y 🔻 🗄 4 🖼 4 4 4 4 6 0	- 🚳 📦 🖾 🖉	0 🥜 🖗 🔳 🔹	• & •	a 3D Swept Mesh	20	Depender	nt							
Simulation Navigator			(FEM) tutorial2017_fe	m.fem ¤ ×			2D Mapped										
Name # tutorial2017_fem.fem.fem.fem.fem.fem.fem.fem.fem.fem.	Display Not Lo		2D Dependent Type Type Select Master and Select Master Face Select Target Face (Match Loops List	Gener d Target Fac (1)	ral Ce	× • • • • • •	3D * 3D Tetrahedral * 3D Hybrid Mesh * 3D Hybrid from Shell Me 2D * 2D Mesh © 2D Dependent * Surface Coat 1D an 2D Dependent	& Sc sh 12 20 8 20) Swept Me lid from Sh) Mapped) Local Rem	iell Me	sh			Master Fac	e		
+ □*Connection Collectors ⇒CSYS ⇒Selection Recipes ⇒Groups		(Filter (Filter (Filter					* 1D M Creates identical		s lement S	ection					P	olygon Fa	ace
- In Fields		(Filter	Mesh Type			^											
Modeling Objects		(Filter	Mesh Type	M Froz	zen Mapı / Cance												

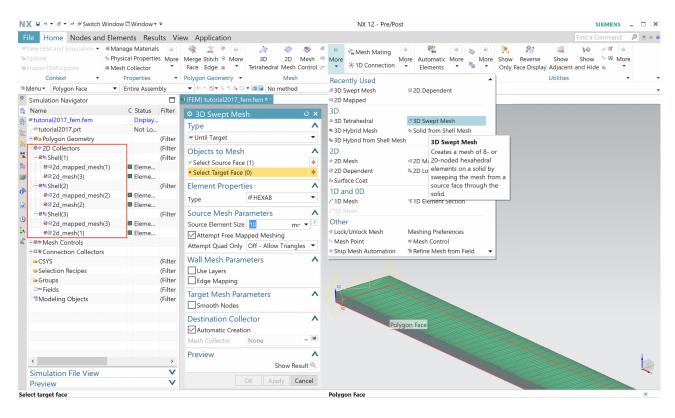
Now, all 3 shells (below "2D Collectors") contain two 2D meshes (the "2D Mapped" and the "2D dependent" ones, see the figure below in section 2.6).

2.6 Create 3D swept mesh

At this stage, when only have 3 shells, each one composed of 2 parallel 2D meshes (upper and lower faces). We will construct the 3D mesh by sweeping the 2D meshes (for the 3 bodies separately). We will now use the mesh constraint along the thickness.

Mesh > More > "3D Swept Mesh" > Type "Until target" > Click the upper face with the 2D mapped mesh, then click in the dialog box to select the target face and click on the lower face of your polygon > OK

You should use *HEXA8* if you have 2D map with *QUAD4* elements. Here again, the "Source Element Size" parameter should have no effect if you had correctly set a constraint of 1 single element on the edges along thickness.



Proceed similarly for all the 3 polygon bodies. You will obtain 3 Solids under "*3D Collectors*", each one containing a 3D mesh. You can verify that you have well a 3D mesh with only one element for the whole thickness and that the top and bottom meshes match such that the elements are vertical prisms. You can again use the static wireframe view.

Name	С	Status	Filter
■ tutorial2017_fem.fem		Display	
□ tutorial2017.prt		Not Lo	
+ № Polygon Geometry			(Filter
+ ≌* Mesh Controls			
+ ≌∻2D Collectors			
■ 🖬 🖄 3D Collectors			(Filter
- ₽ Solid(1)			(Filter
⊿# 3d_mesh(1)		Eleme	
- ₽ Solid(2)			(Filter
■# 3d_mesh(2)		Eleme	
- ⊠tsolid(3)			(Filter
■# 3d_mesh(3)		Eleme	

Do not forget to save your project as yourproject_fem.fem

Remark : Even if the extension changes from the yourproject.prt, it seems it can't have the same name and you should append _fem to the file name.

აx v

•

^

v

Cancel

3 Specify material properties and set the simulation constraints

3.1 Create a sim file

- File > New > under tab "Simulation" select "Simcenter Thermal/Fow" with type "Sim" > OK
- In the field "Associated FEM", you shall select the fem file you just created.
- At this stage under Solver Environment/Analysis type, it appears "Thermal" instead of "Flow" but you
 will be able to change it on the next dialog box.
- After changing the Analysis Type to "Flow", Click OK, as it will still be possible to change all these solution attributes before running the simulation.

🗘 New Simulation 🛛 🖸	X	Solution			
Simulation Name	^	Solution			
sim1.sim		Name	Solution	1	
5001.500		Solver	Simcente	r Thermal/Flow	
Associated FEM	^	Analysis Type	Flow		
FEM tutorial2018_fem 👻 🍃		Solution Type	Flow		
Preview		Flow			
		- Solution De		Description	
Layer Placement	^	Solution Un		Solve Options	
Layer Work	-	- Initial Cond		Run Directory	Current Simulation
		Restart		Flow Solver Selection	Serial Solver
Solver Environment	^	- 3D Flow		Turbulence Model	Mixing Length
Solver: Simcenter Thermal/F	low	Results Opti		Buoyancy	
Analysis Type: There	mal			Solution Type	
Description	^			Solution Type	Steady State
				Advanced	
				Parallel Processing	
OK Cance					
Cance	1				ОК Арр

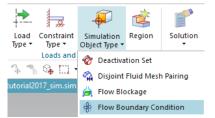
3.2 Set the materials

In the arborescence, expand the "3D Collectors" and double-click every single "Solid". Then, on the left of "Type", click on the symbol \emptyset to \checkmark "Apply Override" for both the "Type (Pure Substance)" and the "Material". On the right of "Material (None)", select "Choose material" and then select the material substance within the catalog : "Water" for this tutorial. Repeat the same for every single "Solid".

+ ⊠≥ Polygon Geometry	Override Mesh Collector Attributes	૨ ×	Override Mesh Collector Attribution	ites し×
+ Override Container	Override Mesh Collector Attributes	^	Override Mesh Collector Attribute	s 🔥
+₩≈2D Collectors	Material	~	Material	
■ Ø △ 3D Collectors + Ø ^L Solid(1) + Ø ^L Solid(2)	Vitrone Pure Substance Apply Override No Override		✓ Type ✓ Matorial ✓ Apply Override	sstance 🔻
+⊠ [™] Solid(3) +⊡*Connection Collectors -□™Fields	Material Orientation Reset to Defaults	v	No Override Reset to Defaults	Choose materia
Z≥ CSYS	OK Apply	Cancel	OK Ar	oply Cancel
Selection Recipes	ок лрру		OK AF	oply Cancel

3.3 Set the boundary conditions

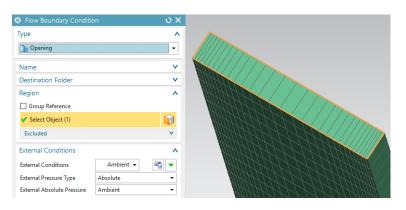
3.3.1 Flow Boundary Condition : Define the inlet and outlet conditions : *Loads and Conditions > Simulation Object Type > "Flow Boundary Conditions"*



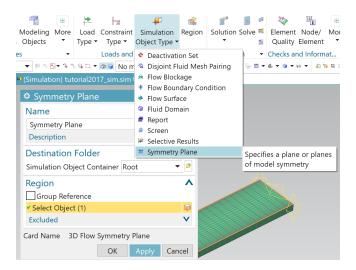
Inlet Flow : to set a velocity (input). In this tutorial, you will set "Velocity" Mode to 0.34 m/s (input velocity at the extremity of long leg of the pipe (on the inlet face)).

Flow Boundary Condition	ა x
Туре	^
🏠 Inlet Flow	•
Name	v
Destination Folder	×
Region	^
Group Reference	
🗸 Select Object (1)	
Excluded	V
Magnitude	^
Mode V	elocity 👻
Velocity	.34 m/s ▼ =

 Opening: to set a pressure (output). Select *External Conditions "Ambient"* pressure at the extremity of the short leg (outlet face).



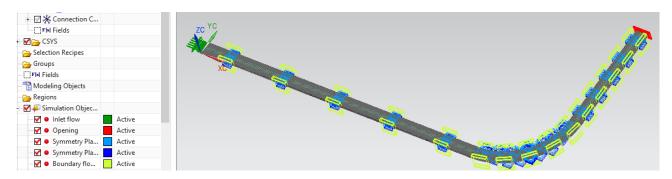
3.3.2 Symmetry Plane : On the upper (and lower) face of the solid, you can impose "Symmetry Plane" to ensure a pure bi-dimensional problem. You can select all faces on one side simultaneously (all 3 faces on the upper surface for example then apply and proceed the same for all 3 faces on the bottom surface), but make sure not to select upper and lower faces together.



3.3.3 Boundary Flow Surface : The faces along the thickness (6 polygon faces in total) represent the walls of the pipe. For them, choose "Flow Surface", with a "Type" of "Boundary Flow Surface". For the "Wall Treatment", select either "Slip Wall" (inviscid fluid) or "Non-Slip Wall" (viscous fluid, zero velocity at the wall), depending on the problem. At the end every single face should have a boundary condition. For laminar flow, you can keep "Non-Slip Wall" but when you use a turbulence model, you have to change it to "Wall Function" (see section 7.1).

ing More tts • Type • Type •	Object Type 👻	Solution S	8		Element	Now Show Only	Reverse Face Display
Loads and Loads and the set of	 Disjoint Fluid Mesh F 			Checks ar		≞ ∷ ▼	
Boundary flow surfation Name Boundary flow surface Description Destination Folder				es wall co ction prop id			
Region Group Reference Select Object (6) Excluded		^ 	J				
Surface Wall Treatment Wall Friction Convection Properties	Wall Function Smooth - With Frict	∧ ▼ io ▼					

You can rename and change the color of the boundary conditions. You can untick the box later in the tree structure (under *Simulation Objects*) for more clarity.



3.4 Set the initial conditions

Loads and Conditions > Constraint Type > Initial Conditions

Setting the initial conditions should speed up the convergence to the solution. In this tutorial, this is not necessary.

3.5 Prepare report for forces on walls

At this point, you will prepare the computation of the forces on the walls (the corner and the legs of the pipe).

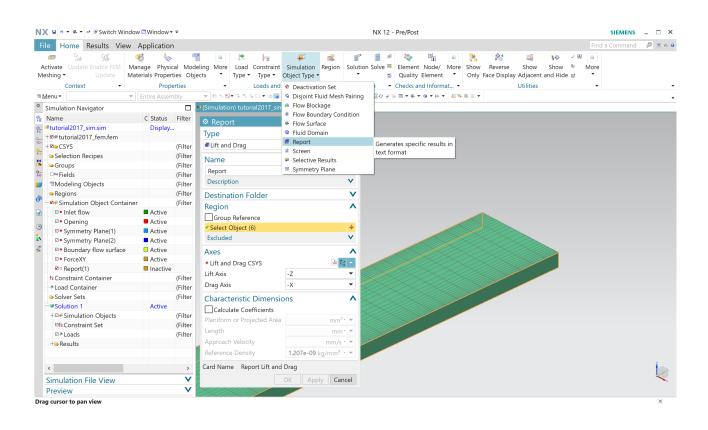
Simulation Object Type > Report > Type "Lift & Drag". Click the surfaces of interest (i.e. the walls, thus the 6 faces along the thickness, as done in the figure below).

In the subpanel "Axes" you should select the coordinate system as previously defined and choose the right orientation for *Lift* and *Drag Axis*.

You can also enter some data to compute directly the aerodynamic coefficient if applicable, like the density ρ (this is not the case for this tutorial).

Do not forget to save your project as yourproject_sim.sim. Even if the extension changes from the yourproject.prt, it seems it can't have the same name and you should append _sim to the file name.

Before running the simulation, check that boundary conditions are active and that all the 3 files (.prt, .fem and .sim) are in the same folder.



4 Solve the simulation

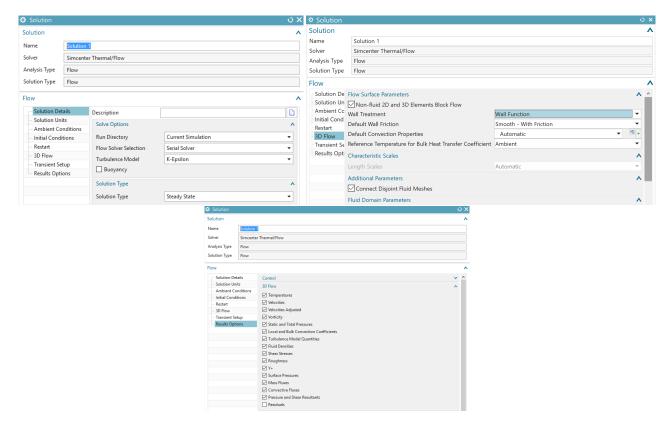
4.1 Set the solution attributes

🗘 Solve		υx	
Solving	Options	^	Solve
Submit	Solve	•	ition
☑ Mod	el Setup Check		
Reso	ve Label Conflicts In Sim Part (None)		
	Edit Solution Attributes		
	Edit Solver Parameters		
Prerequ	isite Solutions Chain	V	
	OK Car	ncel	

Solution > Solve > "Edit Solution Attributes". Make sure Analysis Type is "Flow"

Under Solution Details :

- You can select the *Solution Type "Steady state*" or *"Transient*". In this tutorial, you should start with steady-state.
- You can select the turbulence model. The common one to be used is "K-Epsilon".
- Under 3D Flow, you should enable Use Wall Function.
- Under *Results Options*, you should tick any data field you want to retrieve.



4.2 Set the solver parameters

Solution > Solve > Edit Solver Parameters

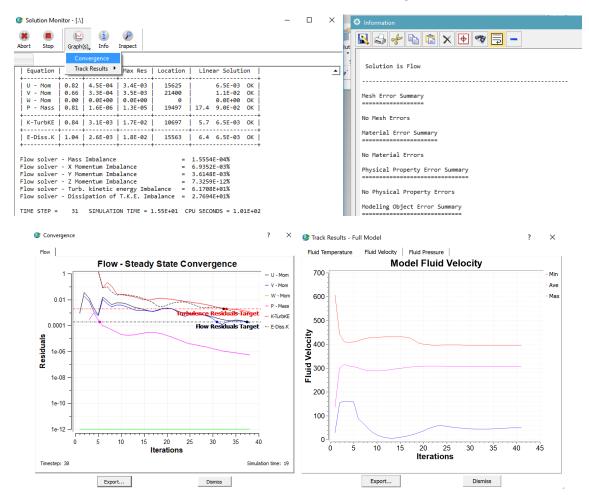
In some cases, you might need to play with the "Steady-State Relaxation Time Step" and the "Time Step" under the 3D Flow Solver tab. It has to respect the Courant–Friedrichs–Lewy (CFL) condition $U\Delta t/\Delta x \leq 1$, with U the velocity (m/s), Δt the time step and Δx the minimal mesh size. You can also change the "Maximum Residuals" of the numerical solver, in order to study the convergence of the solution based on the mesh refinement.

olver		
Simcenter Thermal/Flow		
arameters		
Solver Version		
Version Number 12.0.0, S	ep 12 2017	
Thermal Solver	Convergence Control	
3D Flow Solver	Steady State - Relaxation Time Step	Physical
Radiation Parameters	Time Step	0.5 s •
	Convergence Criteria	RMS Residuals
	Maximum Residuals	0.0002
	Global Flow Imbalance Fraction Option	
	Global Heat Imbalance Fraction Option	
	Global Heat Imbalance Fraction	0.02
		1000
	Steady State - Iteration Limit	1000

4.3 Solve the simulation

Solution > Solve > OK

During the computation, you have two windows : *Information* showing eventual errors or warnings and the *Solution Monitor*. On this window, you can click *Graph(s)* > *Convergence* to check the progress of the computation in real time and/or watch the verbose in real-time (check the residuals). Under *Graph(s)* > *Track Results*, you can also track in real-time min/max/average velocity/pressure convergence. It is important to check that your solution has converged, *i.e.* that the numerical transient phase is finished (the flow needs time to set-up, from rest to the conditions you specified) and that your global quantities reach a plateau.



Export : you can export the graph of convergence/results in .png directly or in . csv to plot it after in Matlab. **Remark :** As the simulation is purely bi-dimensional, the residual for W (velocity along z-axis) is constant at 10^{-12} . This is a quick check for any mistake that could make the flow not purely bi-dimensional. Once, you could run the simulation and it converged, do not forget to save before analyzing the results and trying different configurations.

5 Analyze the simulation results

5.1 Review the verbose

Click Yes to review all the solver verbose. There is very valuable information to understand better the solution and how worked the solver.



5.2 Check the created files

.log: The logging remain available under a yourproject_sim-Solution_1.log file created in the main directory folder.

.html: If you have defined a report with Lift&Drag for instance, an .html file should have been created. **.csv**: The same info is available in .csv file.

.png : Pictures of convergence graphs. You can also export them as .csv to plot them in Matlab.

flow.csc : Physical constants

flow.fli : Results at each nodes (velocity, pressure,...)

flow.gem : Coordinates of nodes

flow.prm : Numerical parameters

flow.job : CFD information

Simcenter Thermal/Flow Report

Length	Tem	peratu	e P	ressur	e For	ce Ve	elocity	Volume	Flow R	ate Ma	iss Flow	Rate	Heat L	.oad	Heat Flu	x Hea	at Trans	fer Coe
mm	C mN/mm*2 mN mm/s mm*3/s kg/s mN-mm/s r					mN/mm-s		mN/mm	1-s-C									
ift Dr	ag																	
.ift Dr	-	LIFT. I	IFT.	LIFT. I	IFT.	LIFT.	DRAG.	DRAG.	DRAG.	DRAG.	DRAG-	SIDE.	SIDE.	SIDE-	SIDF.	SIDE-	PITCH-	PITCH
ift Dr Group	-	LIFT- L X	IFT- Y	LIFT- L Z I	.IFT- Mag. (DRAG- Y	DRAG- Z	DRAG- Mag.	DRAG- Coef.	SIDE-	SIDE- Y	SIDE- Z	SIDE- Mag.		PITCH-	PITCH

5.3 Set-up key measurements for rapid analysis

Once you have solved at least one time the simulation, you can add some key measures of interest. These values are handy for a quick look on the solution when varying different solver parameters or trying different mesh configuration.

Solution tab > Result Measures



Then, click on the lower left corner NEW 📓

Pick up values of interest to evaluate validity and convergence of the solution : like minimum pressure over a surface or maximum velocity over the entire model.

In this tutorial, we will select the min and max velocity and the min and max pressure over the entire domain, and the average pressure at the inlet.

Result Measure	<u>د</u> ک								
Solution	^								
Solution Name	Solution 1								
Input	^								
Result Type	Velocity - Element-Nodal 🔹								
Component	Magnitude 👻								
Coordinate System	Absolute Rectangular 🔹								
Units	mm/s 🔻	Result Me	asure Manager						
Absolute Value		Result Meas	ures						
Operation	^	Solution	Quantity	Component	Operation	Selection Type	Value	Units	
O Minimum	Maximum Mean Average	Solution 1	Static Pressure	Scalar	Minimum	Entire Model	Pmin=-3.0442	Pa	
	() Mican Average	Solution 1	Static Pressure	Scalar	Maximum	Entire Model	Pmax=38.043	Pa	
Model Subset Sel	ection 🔨		Static Pressure	Scalar	Mean Average	Geometry	Pinlet=35.915	Pa	
Entire Model	•	Solution 1	Velocity	Magnitude	Minimum	Entire Model	Vmin=0.04829	m/s	
		Solution 1	Velocity	Magnitude	Maximum	Entire Model	Vax=0.39554	m/s	
Name	^								
Expression Name	Pinlet								
Checking	^								
Requirement	None 🔻	🔣 🔳	G						
	~								
	OK Apply Cancel							C	Clo
									1

5.4 Plot the results

On the very bottom of the three structure, double click on *Flow* under *Results*. Then, under *Flow*, you can access the different data field : velocity, pressure,... To plot the velocity for instance, Click on *Velocity* > *Ma-gnitude*

Solution 1 Active Solution 1 Solution 2	
Image: Second Field Field Second Field Field Second Field Second Field Field Second Field Field Second Field Fiel	
Image: internet	
Image: internet	
Immedia Immedia Immedia	
Image: Flow Imferred Imferred Image: Flow Image: Flow N12-Pe/Port Flow Image: Flow Image: Flow Image: Flow New Section New Section New Section New Section New Section Image: Flow New Section	
Control Pictor C	
Image: Send Window Window P NX 12- PrePart Image: Send State Image: Send State Preve Send State Image: Sen	
Home Results View Application Image: Control Point	
Home Results Wiew Application Prod a Comman Q Image: Comparing the second	
Control Prot Successing Navigation	- 8
Berneller Successing Navigeto Second S	
Sonoth - Sonoth	
Menu No Selection Fill Tanducerey Post View* Post Processing Navigato Edit Post View* Post Processing Navigato Remane * & Adjusted V > Delete * & Pressure on Menue * & Statis Press > New Query Cure * & Statis Press > New Gargh * & Statis Press > New Annotation * & Statis Press > Statis Press * & Statis Press > New Annotation * & Statis Press > Statis Press * & Statis Press > New Annotation * & Statis Press > New Annotation * & Statis Press > Statis Press * & Statis Press > Statis Press <tr< td=""><td></td></tr<>	
Interview	ore ▼
Pot Processing Navigation Image: I	•
Name Image: Construction Image: Construction Image: Construction + Source Market Weight Market Weight Market Weight + Source Market Weight Market Weight Market Weight + Source Market Weight Market Weight Market Weight + Source Mew Graph Source Source + Source Mew Graph Source Source + Source Mew Graphate Source Source + Source Mew Triplate Source Source - Source Mew Triplate Source Source - Source Mew Triplate Source Source <	
 & Adjusted V & Delete & Pressue on & Hide Legend & Hide Legend & Hide Legend & New Ouery Curve & Turbulence & New Graph & New Annotation & Statut & Statut	
 Static Press Static Press Static Press New Query Curve Static Press New Graph New Streamlines Shear Stress New TFile Shear Stress New Annotation Set Result Set Color scheme basis Wentry Set Color scheme basis Honotations Set Scheme basis Set Color s	
 Static Press Static Press Static Press New Query Curve Static Press New Graph New Streamlines Shear Stress New TFile Shear Stress New Annotation Set Result Set Color scheme basis Wentry Set Color scheme basis Honotations Set Scheme basis Set Color s	
+ Source + Sou	
Intel Press New Graph Imported Results Imported Results <td></td>	
Instruction New Graph 966.0 Instruction New J File 37.67 Instruction New Anotation 37.67 Instruction New Anotation 38.73 Instruction Set Result 38.73 Instruction Set Result 38.73 Instruction Set Result 38.60 Instruction Set Result 38.73 Instruction Set Result 38.60 Instruction Set Result 38.73 Instruction Set Result 38.73 Instruction Set Result 38.60 Instruction Set Result 38.60 Instruction Set Result 39.66 Instruction Set Result 39.73 Instruction Set Result 39.66 Instruction Set Result 39.66 Instruction Set Result	
* * Feed Streamlines 366 60 * * * Sheaf Stream 37.67 * * * New Treplate 308.73 * * * New Anotation 279.79 * * * Steamlines 250.85 * * * Steamlines 250.85 * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *	
* Shear Stress New /I File 337.67 * S Yean Stress Mew Annotation 337.67 * S Yean Stress New Annotation 308.73 * S Fluid Teap Set Result 250.85 * S Yelus Elevier Y elevid Stress 21.92 * S Relative Price Sold Results Yei Create Field from Result 260.85 * S Nordar Plats Set color scheme basis 21.92 * S Nordar Plats Set color scheme basis 21.92 * Templates * Set color scheme basis 48.29	
Imported Results New Annotation Imported Results Set Deformation Imported Results A limitate Set color scheme basis Set Result Imported Results Set color scheme basis Imported Results	
308/73 308/73 308/73 279.79 270.79 250.85 220.85 221.92 100.0100 Plots Animate 5 Viewports Contour Plots 5 to clor scheme basis 5 to clor scheme basis 6 Templates	
Imported Results Set Result 279.79 Imported Results Set Deformation 250.85 Imported Results Free Body Results 21.92 Imported Results Free Body Results 100.47 Imported Results Set color scheme basis 100.47 Imported Results 100.47 100.47	
 Fluid Term, St Result St Result St Point Stevents Contour Plots Contour Plots St color scheme basis Termplates St Contour Plots St color scheme basis St	
Styles-Eleg Set Deformation Styles-Eleg Set Deformation Styles-Eleg Identify Relative Pre- Free Body Results Imported Results Free Body Result Styles-Eleg Free Body Result Styles-Eleg Free Body Result Store For View Free Body Result Store For View Set color scheme basis Store For View Set color scheme basis Store For View 109.97 Henglates 109.97 Henglates 48.29	
Solution Plane Solution Solutio	
Relative Pre Absolute Pre Absolute Pre New Create Field from Result New Create Field from Result Animate Set color scheme basis Contour Plots Set color scheme basis Contour Plots Set color scheme basis Animate Set color scheme basis Animate.	
Absolute P File Create Field from Result File Create Field from Result File Create Field from Result Animate Set color scheme basis File Color scheme basis	
Imported Results FM Create Field from Result Viewports Animate Set color scheme basis 32 9 Dott View 108 37 108 37 77 33 48.29 48.29	
Viewports Contour Plots Set color scheme basis Das Etenents Templates Kalimate Set color scheme basis Martin Head 04 Martin Head	
Contour Plots Set color scheme basis Contour Plots Set color scheme b	
← □ Annotations Templates	
< <u> </u>	
< > 48.29	Ť
	-
	-
Preview V [mm/s]	

Post V
 Result
 Color
 Light
 Seeds
 Velo

Edit Plot : At the bottom of the tree, right-click the active *Post-view > Edit Post View*. Or, click directly on the *Edit Post View* button on the top tool bar (*Results* tab). You can choose the *Color Display : iso-lines, arrows...*. For clarity, you can remove the display of the mesh under *Edges & Faces > Edges "Feature"* but it is not recommended to present results during mesh convergence analysis, if results are linked to the mesh.

1	🗘 Post '	View				υx
	Result	Display	Deformation	Legend		
	Colo	r Display S	mooth			•
	✓ Light	ed				
	Edges	& Faces				^
	Edges	Feature	•	Color		
	Faces	External		Color		
		Feature				
	Displa	Wireframe			^	
		None				

Streamlines : To plot streamlines, choose *Color Display "Streamlines"*, under *Seedset* tab, click on *Create*. On the *Seed Set* window, thick *Select Nodal Location* and *Pick "All on face"* on the inlet face for example, to plot streamlines in the whole pipe.

Display Streamlines	liew	υ×	Seed Set	×
tine Display Settings t t t t t t t t t t t t t t t t t t	Display Streamlines	•	Seed Point Select f	from Model 🔻
Remove Seed Point 56 Select All Seed Point 57 Select All Seed Point 59 Deselect All Seed Point 60 Seed Point 61 V	nline Display Settings et city - Element-Nodal	^	 Select Exterior Point Select Point on Cut Plane Pick All on Face Boolean Ope 	+ Auto
			Remove Seed Point 56 Seed Point 57 Seed Point 57 Select All Seed Point 58 Deselect All Seed Point 60 Seed Point 61 Seed Point 61	~

Animation : Click on "Animate" Animate to create an animation of particles on streamlines. You can export it as .gif file.

Animation	X
Animate Streamline	•
Syncronize pulses At Seed Point -	_
Number of Increments 40 Image: Continuous releaseTime Periods 10 Frames per Pulse 5.0000	
Synchronize frame change for all post views Synchronized frame delay (mS) 200 21	:
OK Apply Cance	el

2D graph : Once you have plotted a data field (velocity magnitude for example), you can also create *2D graph* under *Tools* toolbox. In this tutorial, we will plot the velocity profile along the pipe :

- Tools > Create Graph 🔼
- Under X Axis, Define By "Path Length"
- Under Y Axis, Tick "Define By Query Curve"
- Under the *Query curve* tab, click the top right icon to create a new curve.
- Give a name to the curve and select
 - Method "Nodes ID" and pick up "Nodes" with the pen (by selecting the 2 extremities) or "Nodes on Edge" (to select directly all nodes on one edge).
 - Method "Coordinates" and enter manually the coordinates of 2 points. In this tutorial, we will create a curve perpendicular to the pipe at 500 mm after the entry: upper edge point at (500,65,0) and lower at (500,0,0). Finally, enter a high number of points per segment, preferably much higher than the number of elements, N = 100 for instance.
- When back in the graph tab, make sure the slider for *Distance to Mesh* is at the minimum 0.0001 and select the *Projection Vector* to be along z-axis.
- If the "OK" button remains gray and not selectable, try to reverse the direction of the curve by clicking on , below the curve creation button.

😳 Graph	ບ X
Туре	^
On Path	-
Graph Title	^
Velocities along Quer	y Curve
X Axis	^
Define By	Path Length 🝷
Y Axis	^
🗹 Define By Query Cu	rve
Query Curve	^
Curve Name	لم الم
	New
Curve Usage	Project to Element Fac 👻
Distance to Mesh To	olerance A
.0001	1.0000
	_
Projection Direction	n A
Select Vector(1)	
Preview	
Element Nodal Value	Average 👻
Error Handling	^
No Data	Ignore 🔻
	▲
OK	Apply Cancel

Query Curve	×
Name Post Qu Method Node IE	uery Curve (1)
Node IDs : 2 4889, 4768, Clear Reve	Cycle Start Point
ОК	Apply Back Cancel
Ø	Query Curve X
	lame Post Query Curve (2) Aethod Coordinates Pick Nodes
5	oints (one per line) : Show Insert Points 500,65,0 500,0,0 Clear Reverse Cycle Start Point
In	nsert N points per segment: N = 100
	OK Apply Back Cancel

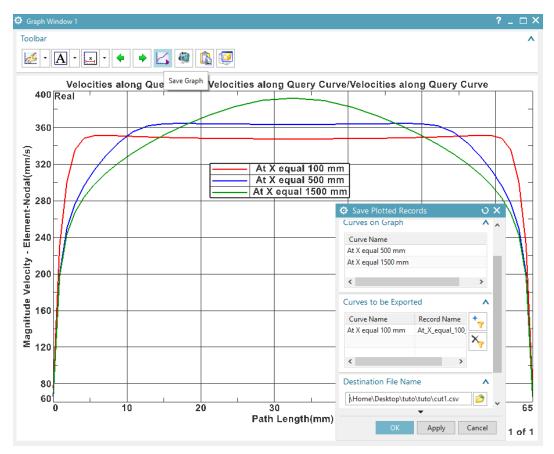
After *OK*, a small pop-up window will then ask you if you want to plot it in a new window or in an existing one, chose to *Create New Window*.

Ø View	/port 🗙
	Create New Window

Plot different 2D graphs on same window : Repeat the same steps to plot the velocity profile at different lengths. When you have your plots under *Graphs* tab (in the tree structure), select all plots that you want, right click *Plot*.

–⊣ { <mark> ∆</mark> } Graphs	
─ ∧ Velocities along	X Delete
∧ Velocities along	
✓ Velocities along	Plot

You get then several curves showing the velocity profile at different lengths from the entry. The turbulent flow profile goes from flat at the inlet (uniform profile at inlet) to a parabolic curve as the boundary layers grow and merge. Within the curve, the profile becomes asymmetric with higher velocity at the inner side of the curve.



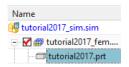
Export 2D graph : On a *Graph Window*, cllick on *Toolbar* and then *Save Graph*. When you have several plots, choose which one to *Export* and select . csv as file format (to import and show in Matlab for example).

A tutorial on how to plot the solution is available here (steps 6 to 9) : https://docs.plm.automation.siemens.com/data_services/resources/nx/12/nx_help/training/en_US/advanced_ sim_tutorial/id741086.html

6 Change the design

In this paragraph, we will show how to test an alternative design, for instance a shorter radius corner.

- First click the top-left icon Return to home Home to exit the result analysis and go back to the simulation navigator.
- In the tree structure, under yourproject_sim.sim/yourproject_fem.fem double-click yourpart.prt two times in a row to open it again. Now you can switch between the part, the mesh and the simulation via the button in the header bar Switch Window.



- When in the *Modeling window* (.prt), select the *Part Navigator* (3rd icon in the left vertical toolbar).

¢	Assembly Navigator
<mark>₿_</mark> Fø	Descriptive Part Name Sections
∎_ ⊦⊚	
ff Pa	art Navigator

- Now you can edit your sketch by double-clicking it. Change the radius from 400 to 70 mm.
- Click Finish Sketch.
- Switch window to go back to the Simulation.
- In the top-left, click Activate Meshing

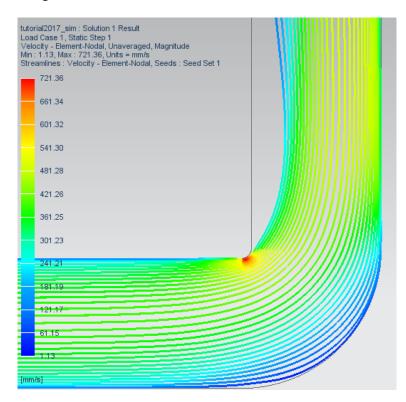


In the top-left, click Update, to automatically update the meshing to the new geometry. Then again in
 Activate Simulation -

the top left Activate the Simulation State

and Solve for this new configuration.

 The same process can be used to change the meshing constraints, if you want to refine the meshing for instance. You can jump between the .prt, the .fem and the .sim files and they should be linked together as you modify them.



Plot the velocity field using streamlines to see the dead zone after the corner.

After loading the solution, Under *Home tab* > *Solution* > *Result Manager* you will see that the values are displayed in red, you can select them and click the bottom *Update* solution to get the new values.

esult Meas	ures						
Solution	Quantity	Component	Operation	Selection Type	Value	Units	
Solution 1	Static Pressure	Scalar	Minimum	Entire Model	Pmin=-281.58	Pa	
Solution 1	Static Pressure	Scalar	Maximum	Entire Model	Pmax=81.572	Pa	
olution 1	Static Pressure	Scalar	Mean Average	Geometry	Pinlet=75.134	Pa	
olution 1	Velocity	Magnitude	Minimum	Entire Model	Vmin=0.0011318	m/s	
Solution 1	Velocity	Magnitude	Maximum	Entire Model	Vax=0.72136	m/s	
i	C						

	R = 400 mm	R = 70 mm
Inlet average pressure	35.9 Pa	75.13 Pa
Maximum velocity	0.39 m/s	0.72 m/s
Maximum pressure	38 Pa	81.57 Pa
Minimum pressure	-3 Pa	-281.58 Pa

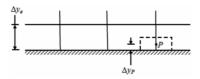
As expected, you can see that the inlet pressure is higher, more than twice what was needed for the large radius. Minimum, maximum pressure and maximum velocity are also higher in absolute values over the domain.

Compared to the analytical prediction ($\Delta p = 31.2 \text{ Pa}$), the pressure drop for the large radius is slightly higher. Indeed, we neglected the entry region (the velocity profile is not fully developped and shear stress are the same along the whole pipe wall). Moreover, the turn at 90° increases the pressure loss and its radius has a strong influence on Δp .

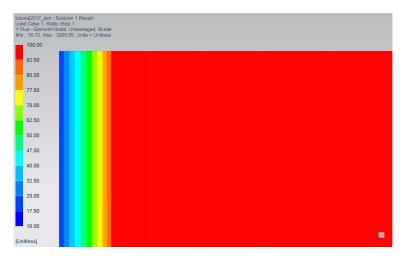
7 Change the turbulence model or solution type

7.1 Turbulence model

The turbulence model can be chosen in *Solve* > *Edit Solution Attributes* > *Solution Details* > *Turbulence Model* ("*None*" if laminar flow). When using a turbulent model and "*Wall Function*" as *Wall Treatment*, you should select the size of the first node adjacent to the wall by looking at the dimensionless wall coordinate y^+ (the law of the wall). Indeed, the first node P must be located in the lower part of the log-law region, i.e. $y_P^+ \in [30, 70]$. The first node Δy_P is located between $\Delta y_e/4 - \Delta y_e/3$ (the size of the first mesh).



This is computed by selecting Y+ and Turbulence Model Quantities (containing YPlus) in Results Options before solving the simulation. The validity of y^+ is crucial since the boundary layer has to be well modeled. You can check it by displaying Y Plus in the results. Then, instead of "smooth" colors, select "banded" to highlight zones of y^+ . Because y^+ is displayed in the whole domain but you are only interested in the values close to the wall between 30 and 70, you can change the boundaries of the legend. On the arborescence, click on Post View, Legend tab and under Value Control, select Specified for Legend Extremes with Min = 10 and Max = 100 for example. You can find more information about it on the documentation : https://docs.plm.automation. siemens.com/tdoc/nx/12/nx_help#uid:xid1128419:index_advanced:xid389654:id627221:xid457884



7.2 Solution type

Depending on the problem, you might solve a transient (unsteady) flow. This can be chosen in *Solve > Edit Solution Attributes > Solution Details > Solution Type > "Transient"*. In this case, you have to select the *Transient Setup* :

- Solution Time Interval : select the physical Start and End Time.
- *Time Integration Control* : select the *Number of Time Steps* or the *Time Step* size (for the CFL condition).
- *Results Sampling* : number of *Results* (at different time steps or *Increment*) that will be stored and shown in the solution.

Solution				×	° F	Post Processing Navigator			
Solution				^	1.00	Name	C	Description	Sta
						tutorial2017_sim			
Name	Solution 1				2	Solution 1		Simcenter Ther	
Solver	Simcenter Thermal/	Flow			27	- ¶≇Flow			Inf
Analysis Type	Flow				10	Increment 1, 0 s			
· · · ·					e_	+ Velocity - Element + Adjusted Velocity			
Solution Type	Flow				H08	+ Pressure on +ve Si			
Flow				^		+ Static Pressure - El			
					0	Total Pressure - El			
	Solution Time Interval		^	^	6	Ha Turbulence Energy			
Solution Un	Start Time	0	s • 🔻			+* Turbulence Dissip			
Ambient Cc	End Time	500	s · 🔻			+ Fluid Density - Ele			
Initial Cond	210 1110	500	5		. N	Hear Stress on +v			
Restart	Time Integration Cont	rol	^		24	+ Roughness on +ve			
3D Flow	Time Step Option	Constant	-			+ ¥ Y+ on +ve Side			
Transient Se		4	c · 🗸			+ Mass Flux - Nodal			
Results Opti	Time Step	1	s · 🔻			H Fluid Temperature			
nesures oper	Results Sampling		^			+ • Y Plus - Element			
	Results	Total Number	•			+ Vorticity - Element + Relative Pressure a			
			•			+ Absolute Pressure			
< >	Number of Results	10		~		+ Increment 2, 0.5 s			
						+ Increment 3, 1 s			
			OK Apply Cano	el		+ Increment 4, 1.5 s			

It is also possible to add *Initial Conditions* from a specified "Uniform" state or "From Results in Other Directory", to start from a previous state, which is already computed.

Solution					૨ ૪
Solution					^
Name	Solution 1				
Solver	Simcenter Thermal/Flow				
Analysis Type	Flow				
Solution Type	Flow				
Flow					^
Solution De	Initial Conditions	From Results in Oth	er Directo	ry	•
Solution Un Ambient Cc Initial Cond Restart	Directory Selection Results Directory				^
-3D Flow	Initial Conditions Selection				^
Transient Se Results Opti	Results to Use for Initial Conditions	At Start Time			•
			ОК	Apply	Cancel

To trigger unsteady phenomena (for example, to force vortex shedding behind a cylinder), you can change the solver scheme : go to Solve > Edit Solver Parameters > 3D Flow Solver > Advection Schemes and change "First-order" to "Second-order (SOU)".

🌣 Solver Parar	neters	ు x
Solver		^
Simcenter Therr	nal/Flow	
Parameters		^
Solver Version		^
Version Number	12.0.0, Sep 12 2017	
Thermal Solve 3D Flow Solve	Advection Schemes	^
Coupled Solve	Momentum	Second-order (SC 🔻
Radiation Para	Limiter	Automatic 👻
	Momentum Limiter Stabilization	General Convectiv 💌
	Energy	Second-order (SO 🔻
	Limiter	Automatic 👻
	Energy Limiter Stabilization	General Convectiv 💌
	Two-Equation Turbulence Model	Second-order (SO 🔻
	Limiter	Automatic 💌
< >	Two-Equation Limiter Stabilization	General Convectiv 👻 🗸

8 Troubleshooting

 A part of the model disappears : In some cases, the 3D view can bug and a part of the model may seem to disappear, as if it were hidden. In this case, you can always go back to any automatic point of view you want thanks to the navigation shortcuts.

6.		- 🖗
6		B
	L	
	0	

2D dependent mesh: As mentioned, you might have some difficulties to create the 2D dependent (not easy to select the Master Face). In this case, check thta you have one 2D mesh for each face and try to change and rotate the view, or you can re-do on the lower face the same procedure as you have done for the upper one to construct the 2D Mapped Mesh.

- Unexpected abortion of the solver :

1. Errors in the Information Window :

Embedded Flow Surface = Flow Surface(1)

There are no valid elements on the underlying geometry.

Make sure all geometry is meshed before performing solution.

That is because you selected an Embedded Flow Surface instead of a Boundary Flow Surface for the walls boundary conditions (section 3.3).

2. Errors in the Solution Monitor :

Processing Lift and Drag Reports...

NX2TMG - Thermal/flow model file builder:

encountered a problem and terminated abnormally.

That is because you select the upper and lower faces (thus faces parallel to the flow) to compute lift and drag instead of the walls in section 3.5.

- Warning in the Information Window :

Boundary Flow Surface = Flow Surface(1)

Unsupported Particle Tracking will be ignored for this solution.

This Warning is not important and will not affect the solution. It concerns only the particle tracking, at the bottom of the window of the Boundary Flow Surface (in section 3.3 for the walls).

9 Extra Resources

NX 12 help menu : https://docs.plm.automation.siemens.com/tdoc/nx/12/nx_help/#uid:index

Many tutorials are available here :

https://docs.plm.automation.siemens.com/data_services/resources/nx/12/nx_help//training/en_US/advanced_ sim_tutorial/index.html?goto=id557841.html

The Simcenter Flow Solver Reference Manual is available here :

https://docs.plm.automation.siemens.com/data_services/resources/nx/12/nx_help/common/en_US/graphics/fileLibrary/nx/tdoc_advanced_simulation/flowrefman.pdf

A community forum for SimCenter 3D simulation is available here : http://community.plm.automation.siemens. com/t5/3D-Simulation-Simcenter-3D-Forum/bd-p/Simcenter_3DSimcenter_forum