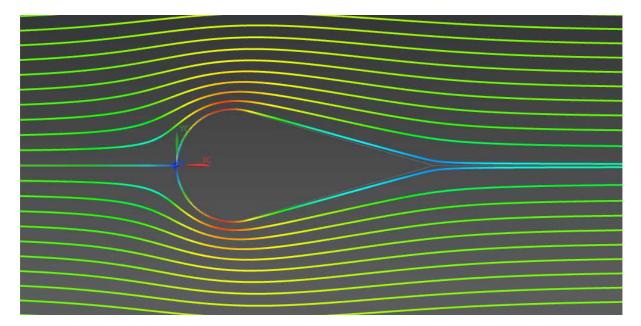
How to set-up and run a 2D Flow Simulation in Simcenter NX11



[Ulg – FSA – Dimitri Arendt] – February 2017 –

Summary

This tutorial explains the workflow to set-up and run a 2D Flow Simulation in Simcenter NX11. 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.

0 Work-Flow

This chapter explains in brief the workflow and main steps needed to perform a flow simulation in SimCenter NX11.

✓ Step1 : Create the fluid domain

- → 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 an arbitrary small thickness.
- ➔ In order 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 yourproject.prt file.

✓ Step 2 : Mesh the body

- → 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 polygon 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

✓ Step 3 : Specify material properties and set the simulation constraints

- ➔ 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".

✓ Step 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.

✓ Step 5 : 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 step3.
- → 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/11/nx_help#uid:xid1128419:index_ advanced:id1245911:id629501

1 Create the fluid domain

1.1 Create an empty model

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

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

NX		Switch W	'indow 🛅 Win	dow 🛪 🗢 NX 11 - Modeling - [model1.prt (Modified)] SIEMENS _	□ ×
File	Home Assemblies Curve A	nalysis Vi	ew Render	Tools Application Vehicle Design Automation OmniCAD for NX Find a Command 🔎 🔳 🐟 🤪 🗕	Ξ×
Sket		Hole	Feature	Image: Chamfer intermediate intermediat	
Ø	Part Navigator				
	Name 🔺	Up to Date			
₽ <u></u> _	🕒 🕒 History Mode				
100 000	+ 🔀 Model Views				
	+ ✔ 🗐 Cameras				
	– ᇋ Model History				
Fo	🖉 🛠 Datum Coordinate System (0)	*			
1				Z	
0				Y	
0				×	
Θ					
<u>[</u> \					
15	Dependencies		v	Ĩ	
3	Details		V		
24	Preview		~		
				Journaling continued	

1.2 Create a 2D sketch

Click "Sketch" in the upper left corner

You can keep the default coordinate axis>OK

Create Sketch	ບ X
Sketch Type	^
🛐 On Plane	
Sketch CSYS	^
Plane Method	Inferred 💌
Reference	Horizontal 👻
Origin Method	Specify Point 👻
✤ Specify CSYS	1 × 1/2 ·
	< OK > Cancel

1.3 Draw the fluid domain

Draw the fluid domain and eventual obstacles using the tools under Direct Skecth : Rectangle, Line, Arc,...

For this tutorial you should start by drawing the following sketch using the line command.

NX	🖬 🤊 • 🕫 🖈 🖻 🖥 • 🛷	🕂 📅 Switch Windo	w 🚺 Window 🕶 🖛		NX 11 - Modelin	g - [model	1.prt (Modified)]	s	EMENS _ 🗆 X
File	Home Assemblies Curve	Analysis View	Render Tools	Application	Vehicle Design Autor	mation	OmniCAD for NX	Find a Command 🔎	🗏 🗠 😮 _ 🗗 🗙
112 117	Sketch In	and the second sec	Linear More Dimension * *	[]] Datum Plane ∭ Extrude → @ Hole	: ㆍ 🧄 Pattern Feature 할 Unite ㆍ 예 Shell Feature	1		More Surface	Assemblies *
雪	enu 👻 No Selection Filter 👻 Entire Ass	embly 👻 🟥	🗠 🕂 🔹 📬 E] 🔹 💿 💿 🐼	1124~	$+ \odot$	0+/3	0 🖬 0	· 🔲 • 📦 • 🕪 • 🖕
Ø	Part Navigator								
	Name 🔺	Up to Date							
₽ <u></u>	🕒 🕒 History Mode								
100 C	+ B Model Views								
-	+ V 🚳 Cameras								
R	- 🍃 Model History								
F@	Datum Coordinate System (0)	¥							3
"		*							601
0									
٥			ľ						
٩						-			-
•				×					•
-					1	.85,2			
K	Dependencies	v	1						
37	Details	~	ĕ —► X						
E.e	Preview	v							
Select	objects and use MB3, or double-click an objec	t		Sketch is fully	constrained with 4 auto	o dimension	ns		

You will see that grey dimensions appear automatically in such a way to fully define your sketch. These are auto-dimensions. You can double-click some and adjust the value to make them constraints, they will appear in dark blue color. You can add constraints by selecting a segment and then choosing "horizontal dimension" or "vertical dimension".

You can also add more complex constraints, like parallelism, same length, perpendicularity, etc.. by selecting successively different elements and then choosing suggested constraints.

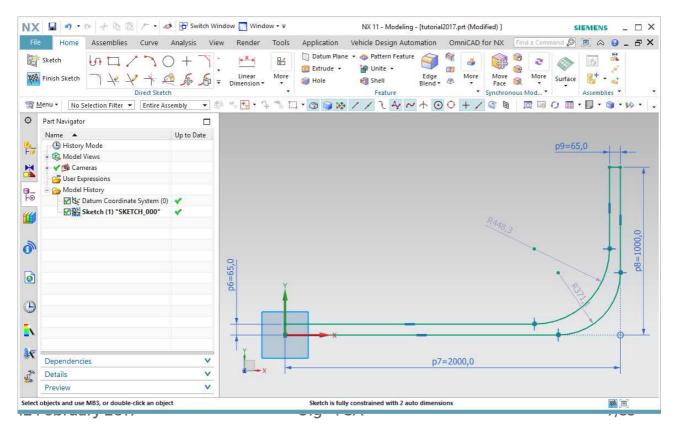
For instance, if you want a segment to be symmetrically located with respect to the coordinate axis, you select first the mid-point of the segment (with left mouse click), then the x-axis arrow (no need to hold down CTRL or SHIFT key) and then a suggestion of constraint automatically appears "point on curve" and you can click it. You can also find these constraints under *Direct Skecth>More>Skecth Constraints*

For this tutorial you should have the following constraints.

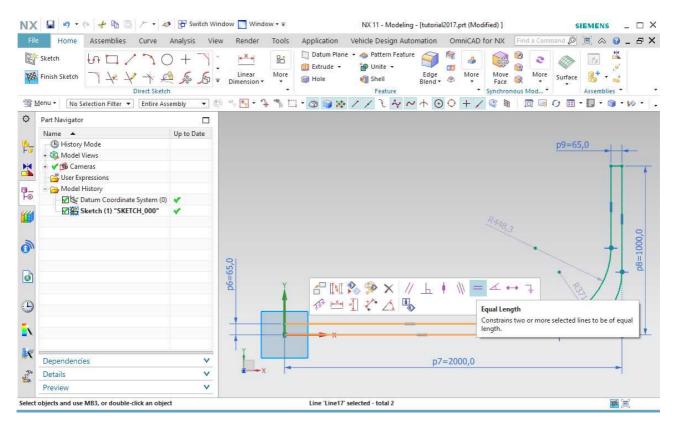
X	📱 🤊 • ભ 🕂 🗛 🖪 🖪 •	🧈 📅 Switch V	Vindow 🔝 Wir	dow + ∓		NX 11 - Modeling	g - [model	1.prt (Modifi	ed)]		S	SIEMEN	s .	- 🗆	1 >
File	Home Assemblies Curve	Analysis V	iew Render	Tools	Application	Vehicle Design Autor	nation	OmniCAD f	or NX	Find a Com	mand 🔎		0.	. 8	3
22	Sketch Finish Sketc	〇 +	÷ Linear ▼ Dimension	More	[] Datum Plane ∭ Extrude → @ Hole		Edaa	More	Move Face	More	- Junace	-	•		
P M	Direct Si ₫enu ▾ No Selection Filter ▼ Entire	etch Assembly 👻	ti + + +		7 - 0 - 10	Feature	+ 0			ous Mod T	0 🔳	Assemb		- A	
			190 LA 14	+ 11 L	-1 - 🕼 🔰 🐝	// (4~	Τ 🙂		<u>ज</u> स्व	1021				192 T.	2
10	Part Navigator														
3	Name Bergen History Mode Content With the second s	Up to Date									<u>p9=65</u>	5,0		-	
1	User Expressions													1	Ī
0	- 🔁 Model History														l
۲	Datum Coordinate System														1
1														0	
			Q											p8=1000,0	
			p6=65,0	Y										đ	ĺ
)				1	-										
-								_					•	_	ł
5	Dependencies	~	Ĭ.	_			p7=	2000,0							
	Details	v	é	x									1		
	Preview	v													
	objects and use MB3, or double-click an o				Sketch is fully								V #	-1	

Ideally when you are done with your sketch as here-above, it shouldn't contain any grey dimensions (auto dimensions) but only dark blue constraints (px=...) set by you. This means the sketch is fully constrained. The constraint status also appear in the status bar (bottom of the window).

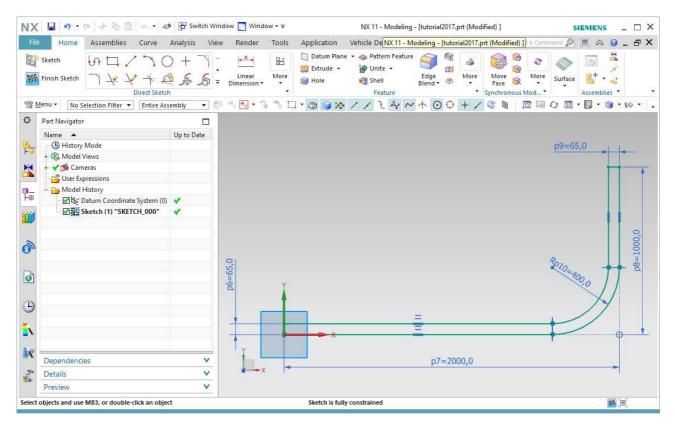
The next step is to make a round corner. Select the "filet" tool under "Direct Sketch" and then successively the two inner legs of the corner. Repeat with the two outer legs. You should have the following sketch with two additional degree of freedom, being the two arc radius.



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.



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.



TIP : 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

TIP : How to navigate

- 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 short-cut SHIFT+F8 to get back to the top view.
- Other usefull shortcut are available when holding down the right mouse button.

1.4 Finish the sketch

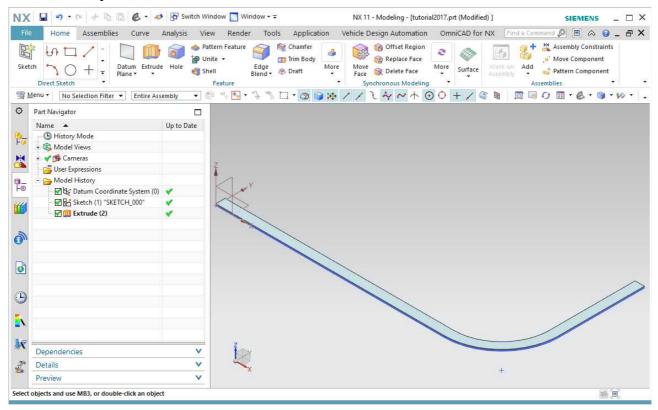
When you are done with the sketch, *Click Finish Sketch in the upper left*

1.5 Extrude the part

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

NX	🖬 🤊 · 🤄 🕂 ት 🛱 🛄 · 🤣	🗗 Swit	ch Window 📃 Wind	iow + ∓		NX 11 - Modeling - [tutoria	al2017.prt (Modifi	ed)]	SIEMENS	- F	⊐ ×
File	Home Assemblies Curve A	nalysis	View Render	Tools /	Application	Vehicle Design Automation	OmniCAD fo	NX Find a Com	mand 🖉 🗐 🐟	0 _ d	× R
Sketch	Direct Sketch + Plane + +	H Sect	Select Curve (8)	~ ~ ~	0 × ^ 10	Goffset Region Generation Generation	More Surface	Work on Add Assembly Add Ase \sim \circ +	Move Componer Move Componer Pattern Compon semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies Componer Semblies	nt	
0	Part Navigator	Lim	ection ite		~						•
*** *** ***		Up Star Dist End Dist Cost Boo Boo	t ance Open Profile Smart Vo Ilean Ilean (None) ect Body (0)	Value V	mm • mm •				65 	1000	2
 Image: A = 1 Imag	Dependencies		< 0K>	Apply	Cancel		2000			d 10	
	Details Preview		v v							4	
_`* -	ection geometry		•		No Boolean	will be performed.			1	va (=)	

By rotating the view with the middle mouse button you can verify that you have now a 3D solid body.

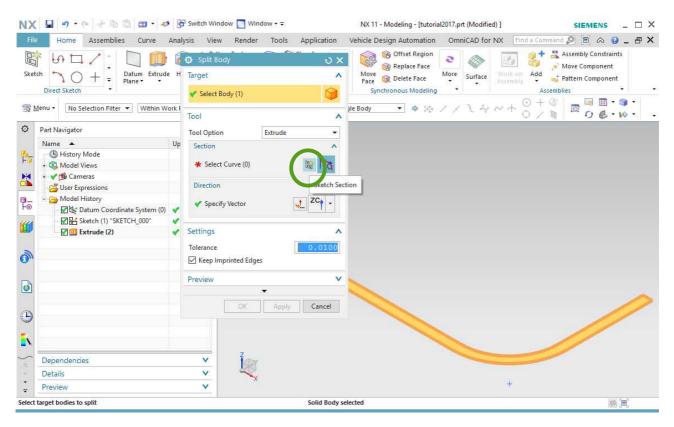


1.6 Split the body to prepare the meshing

In order to be able to make a consistent meshing with quadrilateral elements, the fluid domain has to be divided in simple geometries: triangles or quadrilaterals, possibly with curved edges.

Under Feature toolbox>More>Trim>Split body

Select your body and then under Tool>Section>Select Curve>Click on Sketch Section



Just click OK on the next dialog box to select the same system coordinate (CSYS) you already used.

Proceed similarly as for the initial sketch. For this tutorial, 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, ideally by having your new elements 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.

TIP : For boundary layer on complex geometry a nice tool to experiment is the "offset curve".

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

Make sure to tick the box for "Keep Imprinted Edges" under Settings and click OK.

How to set-up and run a 2D Flow Simulation in Simcenter NX11

NX F		æ	Switch Window 🔲 Wi	ndow	NX 11 - Modeling - [tutoria	l2017.prt (Modified)]	SIEMENS _	□ ×
File	Home Assemblies Curve	Anal	ysis View Render	r Tools Application	Vehicle Design Automation	OmniCAD for NX	Find a Command 🔎 🗐 🐟 🥹 🗕	ъ×
Sketch Dir	Image: Sketch Image: Datum Extrude Plane * Image: Datum Extrude Plane * Image: Datum Extrude Plane * Image: No Selection Filter * Within Works		 Split Body Target Select Body (1) Tool 		Move Face Replace Face Synchronous Modeling		Assembly Constraints	
Ø Part	Navigator		Tool Option	Extrude 💌				
	me A History Mode Model Views Source Spressions Model History Model History	Up	Section Section Select Curve (2) Direction Specify Vector Settings Tolerance Keep Imprinted Edg Preview OK V V V	Apply Cancel			÷	
Select sectio	on geometry							

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.

For more complex geometry, you can repeat the operations as many time as needed to be able to create a sound mesh later on.

Make sure you save your model as a yourproject.prt file.

Sketch tutorials can be found here :

https://docs.plm.automation.siemens.com/tdoc/nx/11/nx_help#uid:id1251042

https://docs.plm.automation.siemens.com/tdoc/nx/11/nx_help#uid:xid1128417:index_ sketcher:id188016:id771117

Sketch video examples can be found here :

https://docs.plm.automation.siemens.com/tdoc/nx/11/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*

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.

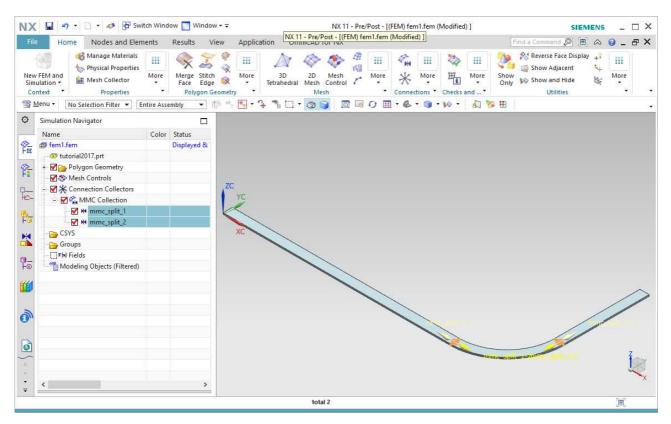
Select Solver "Simcenter Thermal/Flow" and Analysis Type "Flow".

Click OK.

🔁 New FEM			υx
FEM Name			^
fem1.fem			
CAD Part			^
Associate to N	/laster Part		
Part tp2016		•	
Idealized Part			^
Create Ideal	ized Part		
Bodies			^
Bodies to Use	All Visib	le	•
Polygon Body F	Resolution	Standard	•
Geometry			^
	Geometry	Options	
Solver Environr	nent		^
Solver	Simcente	r Thermal/Flow	•
Analysis Type	Flow		-
Default Cyclic S	ymmetry (Cylindrical CSYS	^
Use Cyclic C	SYS if Defi	ned	
Specify CSYS		1. A.	4 (4) + (4)
Mesh Morphin	g		^
		1	۸
Save Full Mor		1	^
Mesh Morphin		1	^
Save Full Mor			^

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. Check the MMC (Mesh Mating Conditions).



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.*

TIP : When you are sure about the connections, you can hide the symbols by unticking the Connection Collectors in the arborescence tree to keep the visualisation more clear.

TIP : You can change the colour of your polygons by double clicking the small color box next to it in the arborescence and rename them to make identification easier later on.

2.3 Add the meshing constraints on the body edges

Mesh Control>chose in the dropbox Number on Edge/Size on Edge/Biasing on edge >Select all edges for which you want to apply the constraints > Click the preview button > OK.

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 start, end of edge or middle of edge) and it varies from edges to edges depending on the edge natural direction.

In this tutorial 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.

How to set-up and run a 2D Flow Simulation in Simcenter NX11

X	📓 🦻 🔹 🖑 🐨 🐼 🔂 Switch Wind	ow 🔄 Window - 🖛	Ν	X 11 - Pre/Post - [(FEM) fe	m1.fem (Modified)]	SIEMEN	s _	
File	Home Nodes and Elements	Results View	Application OmniCAD for I	٩X		Find a Command 🔎 🔳 🖉	0_	в×
New F	Manage Materials More Manage Materials More Manage Materials More More More More More	Merge Stitch 5-co 5dge 0 X n Geometry	 Tetrahedral Mesh Cor 	sh More K	More More	Show Only Wo Show and Hide Utilities		
	Density Types	A] 🕄 🐾	🗄 • 😘 🐃 🗔 • 🚳 🌍 [No method 🔻 🖣		田 🕺 🕼 🔹 🕪 - 📦 - 💩		
}	Biasing on Edge	- I						
H	Selection	<u>∧</u> 3						
-	✓ Select Targets (4)							
	Biasing on Edge	^	YC					
	Bias Origin Center of Edge	- <	\leq					
2	Number of Elements	30 🗘 🕺	C					
	Bias Ratio 1.12	•						
1	Auto Size	9						
5	Preview	<u> </u>						-
2	Automatic				<	/		
1	Mutomatic	0						
	OK Apply I	Cancel						
1	OK Apply I	Lance						
1							5	z
-								By
								X
	<	>						
act of	bject to define mesh control							

Remark : A matching edge between two polygons should be specified once only.

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.

X 🖬	🤊 🔹 🛃 🔹 🤣 📅 Switch Wi	ndow 📘 Window 🕶 🖛	NX 11	I - Pre/Post - [(FEM) fem1.fem	(Modified)]	SIEMENS	-	
File H	Home Nodes and Elements	Results View App	olication OmniCAD for NX		Find	a Command 🔎 🔳 🐟	0 _	8×
New FEM and Simulation * Context	d Manage Materials	Face Edge 🔍 🕶 Polygon Geometry	re 3D 2D Mesh Tetrahedral Mesh Control		More Show Only W	Reverse Face Display + Show Adjacent + Show and Hide Utilities	More	
<u>M</u> enu ▼	Polygon Edge 🔹 Entire A	ssembly 💌 👘 🔧 🏪	• 🗘 🦄 🗔 • 🚳 🕥 No	method 👻 🎐 🧕	0 🖩 • 🗞 • 🌒 •	田愛し、		
Simulat	tion Navigator D Mesh Control	⊓ ບ×						
	Density Types	^						
in the second seco	Size on Edge	*						
S	Selection	^						
	🗸 Select Targets (4)	¢ v	с					
5 S	Size on Edge	^ S	The second s					
1 📫 L	Location on Edge Overall	→ XC	Contraction of the second s					
	Element Size 10	mm 🔻	A DECEMBER OF THE OWNER					
ء ا	Auto Size	1		and the second se				
1 P	Preview	^		And the second distance of the second distanc			MATHMAN	1161
E E	Automatic	0					COLOCULAR .	
						- Lastan		
3	OK Apply	Cancel						
~		1					1	Z
2								
<		>						
ect object to	o define mesh control							

NX 🖬	🤊 • 🐟 • 🛷 🎛	Switch Window 📃	Window • ∓			N	X 11 - Pre/Pc	st - [(FEM) fe	m1.fem (Modi	fied)]			SIEMEN		□ ×
File + New FEM and Simulation + Simulation + Simulation + Simulation + Image: Simulation + <	Home Nodes and El Manage Material Physical Properties Polygon Edge Polygon Edge Mesh Collector Properties Polygon Edge Selection Selection Size on Edge Location on Edge	lements Result Is es More + Face	s View	Applica More	D Tetrahedral	INICAD for N 2D Me Mesh Con Mesh Con Mesh	IX sh trol C M	III FM	III × More ∰ ctions → Cheo	More tks and Y	Show Only 1	Steverse Fa	Ce Display + cent + Hide &	9 _	₽×
	OK	Apply Can	cel >								muluus				Ŀ,
	o define mesh control					1 object	selected								

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

Using *Mesh Control>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.

١X		7	• 😻 • 🛷 🎛 s	witch Win	dow 🔲 V	Vindow 🕶	2			NX 11	1 - Pre/F	ost - [(FEM) fen	n1.fem (Modified	0]			SIE	IENS		□ ;
		Hon	me Nodes and Ele	ements	Results	View	Appli	ation O	mniCAD	for NX							FI	nd a Commar	d 🖉 🔳		0 _	8>
New Simu	FEM a ulation	and n∙ ▼	Manage Materials Physical Propertie Mesh Collector Properties			Stitch Edge	More	3D Tetrahedra	2D al Mesh Me		単同で	More	Connec	More •	Checks	More	Show	Neverse f	ljacent	v + + ₩	More	
<u>₿</u> <u>M</u>	lenu -	•	Polygon Edge 🔹 👻	Entire Ass	embly	▼ #	*	° ∔ "∖ 🖂	• 🐲 🕯	No I	method		-	0	0	- 26	6 • 6	- 10 -	1 74			
¢	Simu	lation	n Navigator Mesh Control			л v x																
2			nsity Types		3	~																
-щ	(and)	-	Number on Edge																			
2	+	-	, Number on Edge			<u> </u>																
1000	E.	Sele	ection			^																
		4	Select Targets (8)																	_	-0	0
	+-1	Nur	mber on Edge			^												_	/			*
1	1	Nur	mber of Elements	1		: >	_										/					
	-	Aut	to Size	ŀ		1	R							_								
4		Pre	view			^								-	-							
Ð			Automatic			0																
9			өк	Apply	Cano	el																
3																						
2																						Z_
24 - 22 - 22																						~
	۲					>																
	10000		efine mesh control						1.0	bject sele	stad											_

TIP : When setting mesh constraints on edge, you should rotate the view to an out of plane view angle to differentiate between the edges on the lower face and on the upper face. You should constraint only the edge on one face, for instance all edges on the upper plane.

TIP : For more complex geometry, you can set the view to 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

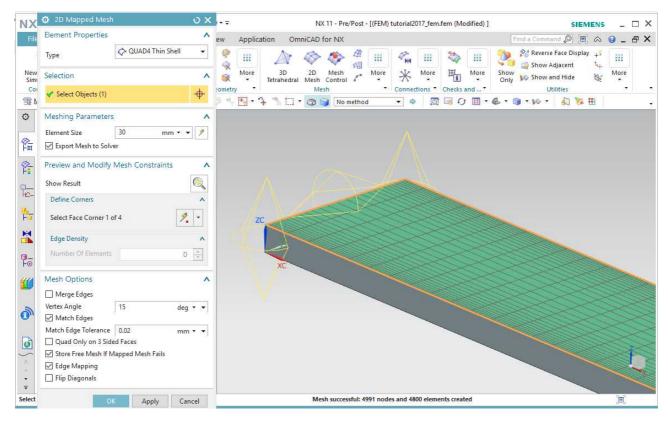
You can now mesh the face by using 2D mapped mesh. Off course you should mesh the plane on which you defined the mesh constraints. From experience, it is recommended to map the face one by one.

Mesh>More>2D Mapped Mesh>Click the face where you set the edge constraints

You can select either QUAD4 or TRI3 elements. In this tutorial we select quadrilateral 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.

You can click the icon next to "show results" to get the preview before clicking OK.



Mesh all the other polygon faces of the plane similarly.

In more complex geometry, make sure not to over-constrain the mesh. 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 hereabove). 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 top and bottom faces of the solid.

For every single 2D mesh, a dependant 2D mesh on the opposite side has to be created.

Mesh> More > 2D dependent> Select the master face with the mesh > Select the corresponding face on the opposite side > OK.

Repeat the process for all 2D meshes. You might try to map more than one polygon face at a time to speed up the process if you have many of them but it is not always possible.

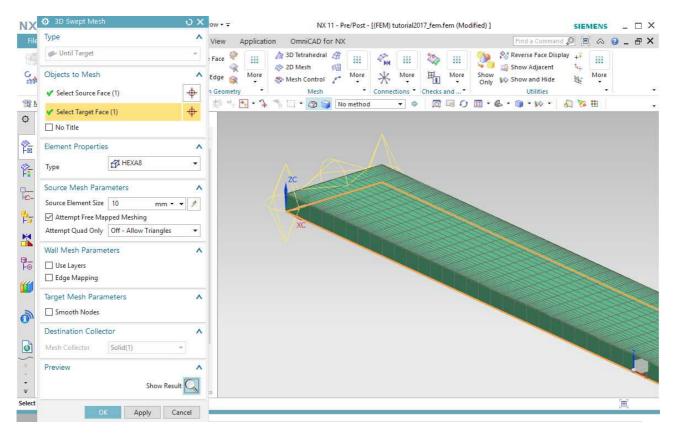
2.6 Create 3D swept mesh

The 3D mesh can now be created by sweeping the 2D mesh.

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

You should use HEXA8 if you have 2D map with QUAD4 elements.

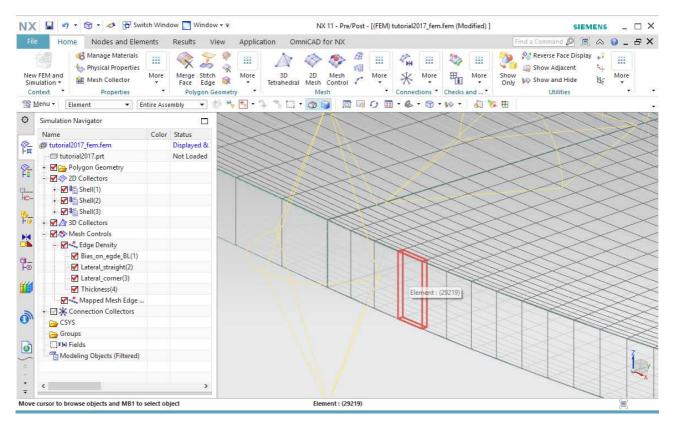
Here again, the "Source Element Size" parameter should have no effect if you correctly set a constraint of 1 single element on the edges along thickness.



You shall have only a single element on the thickness for a pure bidimensional simulation.

Proceed similarly for all the polygon bodies.

Verify that you have 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 wireframe view.



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.

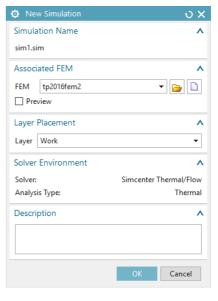
3 Specify material properties and set the simulation constraints

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.



Solution				
Name	Solution 1			
Solver	Simcenter 1	Thermal/Flow		
Analysis Type	Flow			•
Solution Type	Flow			•
Flow				
Solution De	tails	Description		
- Solution Un		Solve Options	L	^
- Ambient Co		Run Directory	Current Simulation	•
Restart	luons	Flow Solver Selection	Serial Solver	•
- 3D Flow		Turbulence Model	Mixing Length	
Transient Se		Buoyancy	mang cenger	
Results Opt	ions			
		Solution Type		^
		Solution Type	Steady State	•
		Advanced		^
		Advanced Parameters (0)		
		Generic Entities (0)		
		Run Job in Foreground		44
		Parallel Processing		^
		Run Solution in Parallel		

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.

3.2 Set the materials

In the left arborescence, expend the 3D meshes and double-click every single mesh (1) collector; then double click next to propert\ytype (2) and then select the material substance within the catalogue (3): water for this tutorial.

NX	📓 🤊 • 🗋 • 🛷 🔂 Swite	h Window 🗾 Window 🕶 🖛	NX 11 - Pre-	/Post - [(Simulation) sim1.sim (M	lodified)]	SIEME	NS _	
File	Home Results View	Application OmniCAD for NX				Find a Command 🔎 🔳 🗸	≈ 🕜 _	₽×
Disp	Change layed Part + ontext + Properties	More Load Loads and Conditions	t Type - Solution Solve	Checks and Information	e Show Rever Only Face Dis		More	
0	Atenu 👻 🚽 🐨 🐨	thin Work Part O 🔹 🖏 🦏 🔩 👻	• "\ [] • @ 🔂 🗖 [🗟 🗘 🔢 • 💩 • 📦 • 🕪	• 🛃 🐼			•
~	NAMES OF A DESCRIPTION OF A	15-76						
A	Name	Color Status Displayed A						
FI	and a straight straig	Override Mesh Collector Att	ributes	2 X				
	 ✓ Intronal 2017, fem.fem Intronal 2017, prt ✓ Mesh Controls ✓ Mesh Controls ✓ Polygon Geometry ✓ 20 Collectors ✓ A 3D Collectors ✓ A 3D Collectors ✓ A 50 C	Override Mesh Collector Attrib Override Mesh Collector Attrib Material Vype Material Reset to Defaults Property Type Material	butes Pure Substance	3				
0	Regions Simulation Object Container Constraint Container		Apply Cance	el			Ĩ	da.
4 4 4	 Load Container Solver Sets 	~						X
	lesh Collector Attributes							

Repeat the same for every single 3D mesh.

3.3 Set the boundary conditions

Define the inlet and outlet

Loads and Conditions>Simulation Object Type>Flow Boundary Conditions>

- Inlet Flow : to set a velocity (input)
- Opening : to set a pressure (output)

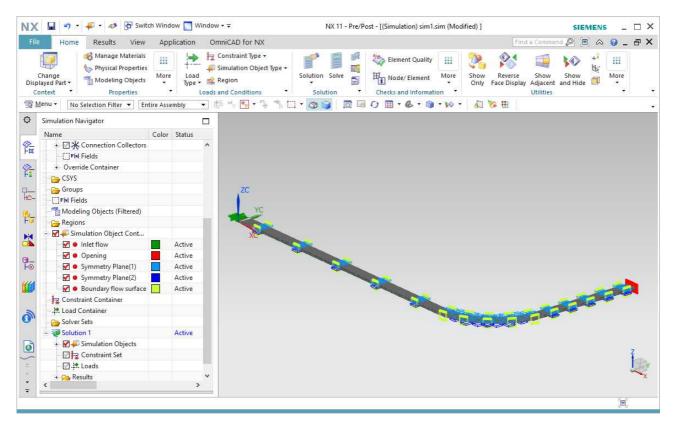
In this tutorial, you will set 0,34m/s as input velocity at the extremity of long leg of the pipe and ambient pressure at the extremity of the short leg.

On the top and bottom face of the solid, you can impose symmetry plane to ensure a pure bidimensional problem. You can select all faces on one side simultaneously, but make sure not to select upper and lower faces together.

Loads and Conditions>Simulation Object Type>Symmetry

On the remaining lateral faces, you can select either *Flow Surface\Boundary Flow surface\slip wall or non-slip wall*, or symmetry depending on the problem. At the end every single face should have a boundary condition. When you use non-slip boundary surface, make sure to tick "Wall function".

You can visualize the boundary conditions and untick the box later in the arborescence 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 faces.

At this point, you will prepare the computation of the forces on the corner and the second leg of the pipe.

Simulation Object Type > Report > Type = Lift & Drag

Click the surfaces of interest. For this tutorial, select the 6 polygon faces as shown here-below.

How to set-up and run a 2D Flow Simulation in Simcenter NX11

ivate ivate hanage Physical Modeling Manage Physical Modeling Materials Properties ForceXV Materials Properties Materials Pro	and Conditions	Solution Solve Solution View Solution	Element Quality		Find a Command 2 Reverse Show ace Display Adjacent Utilities	Show and Hide	+' ⊌≤	HII More
ivate ivate Manage Physical Modeling More Materials Properties ForceXV Manage Physical Modeling More Materials Properties Materials Properties Mat	Simulation Object Type + Region and Conditions +	Solution Solve	Hill Node/ Element More Checks and Information		Reverse Show ace Display Adjacent Utilities		5	More
ForceXY C						-		
orceXY	Î					日 🖗 田		
Vescription V								
stination Folder								
nulation Object Container 🛛 Root 👻 💋								
gion 🔨								
Group Reference Select Object (6)								
xcluded Y								
iciudeo 🗸 🗸								
es 🔨		<u>.</u>						
Lift and Drag CSYS							-	
Axis X 👻		ALTERNA DE LA COMPANY				-		
ag Axis Y 👻			State of the second second	in manual	C.C.C. Statement			
aracteristic Dimensions								
Calculate Coefficients	v							7
OK Cancel								1
								Y
< >								

You should select Lift axis to be X and Drag axis to be Y.

In the subpanel "Axes" you should select the coordinate system as previously defined and chose the correct axis for Drag and Lift.

You can also enter some data to compute directly the aerodynamic coefficient if applicable. This is not the case for this tutorial.

Do not forget to save your project as yourproject_sim.sim

Remark : 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.

4 Solve the simulation

4.1 Set the solution attributes

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 tutotial, 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 Option, 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 "Time Step" under the 3D Flow Solver tab.

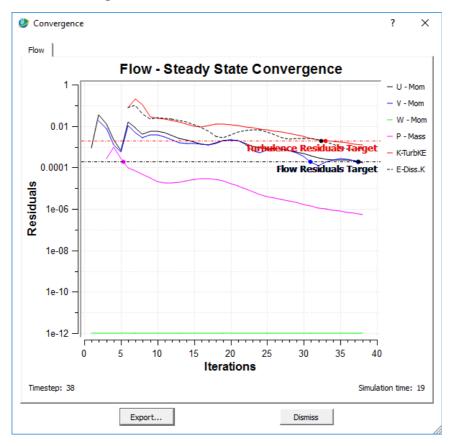
You can also change the Maximum residuals. In order to study the convergence of the solution based on the mesh refinement, you could put a smaller "Maximum Residuals".

4.3 Solve the simulation

Solution>Solve>OK

During the computation, in the Solution Monitor window, you can click Graph>Convergence to check the progress of the computation in real time And/or watch the verbose in real-time.

Under Graph>Track Results > You can also track in real-time min/max/average velocity/pressure convergence.



Remark : As the simulation is purely bidimensional, 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 bidimensional.

Once, you could run the simulation and it converged, do not forget to save before analysing the results and trying different configurations.

5 Analyse the simulation results

5.1 Review the verbose

🔮 Rev	iew Results			×
2	The solution	is completed. Do	you want to review all	messages now?

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

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

5.2 Check the created files

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

🗋 Simcer	nter Therma	al/Flow	<												<u> </u>	— C	× c
$\leftarrow \rightarrow c$; 🛈 file	:///C:/Us	sers/Dir	mitri/Drop	box%20(F	Personne	lle)/Persc	o/Jobs/ULg/M	IECA0025	/TP-nx/tutoria	al2017/	'tutoria	al2017_sir	m-S ☆	a 🔀		<i>J.</i>
Applicatio	ons 📙 D)im 📃 A	Astro	Fleye	Daily	🗧 Google	e Docs 🕻	🕽 GitHub 🛛 💓	TweetDeck	F Fleye 💆	🖌 Audie	nce 🤇	🗿 Helpdes	sk : Fleye	, To read	📙 Outils	s »
MA I		R				n: C:/User Solutior Dimitri : MAYA 12-Feb	rs/Dimitri/D n_1 -2017 - 09	er T Propbox (Person :31:18							•		
Inits																	
Length			re Pr	essure	Force	e Velo	city	Volume F Rate	low	Mass Fl Rate	ow		eat bad	Heat Flux	He	at Tran Coeff	
Length		eratui c		essure	mN	e Velo	city		low		ow	Lo			He		
Length mm _ift Drag	g	С	n	nN/mm^2	mN	mm	n/s	Rate mm^3/s		Rate kg/s		Lc mN-	mm/s	Flux mN/mm-s		Coeff mN/mm-s-	с с
 .ift Drag	g		n	N/mm^2	mN	mm	n/s	Rate		Rate kg/s		Lo mN-	mm/s	Flux mN/mm-s		Coeff mN/mm-s-	c · SIDI
mm _ift Drag	g	с _IFT- <mark>I</mark>	n	N/mm^2	LIFT- I	mm	DRAG	Rate mm^3/s	DRAG	Rate kg/s	- DR/ Co	Lo mN-	mm/s	Flux mN/mm-s	SIDE-	Coeff mN/mm-s-	c SIDE Coe

The convergence graphs are also available afterwards as png image in the main directory.

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>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	0 X
Solution	^
Solution Name	Solution 1
Input	^
Result Type	Velocity - Element-Nodal 🔹
Component	Magnitude 👻
Coordinate System	Absolute Rectangular 👻
Units	m/sec 💌
Absolute Value	
Operation	٨
O Minimum	Maximum O Mean Average
O Minimum Model Subset Sele	,
0	,
Model Subset Sele	,
Model Subset Sele	,
Model Subset Sele Entire Model Name	ection

Solution	Quantity	Component	Operation	Selection Type	Value	Units
Solution 1	Static Pressure	##11Scalar	Minimum	Entire Model	Pmin=-3.0442	Pa(N/m^2)
Solution 1	Static Pressure	##11Scalar	Maximum	Entire Model	Pmax=38.043	Pa(N/m^2)
Solution 1	Static Pressure	##11Scalar	Mean Average	Geometry	Pinlet=35.915	Pa(N/m^2)
Solution 1	Velocity	Magnitude	Minimum	Entire Model	Vmin=0.04829	m/sec
Solution 1	Velocity	Magnitude	Maximum	Entire Model	Vax=0.39554	m/sec
×.	G					>

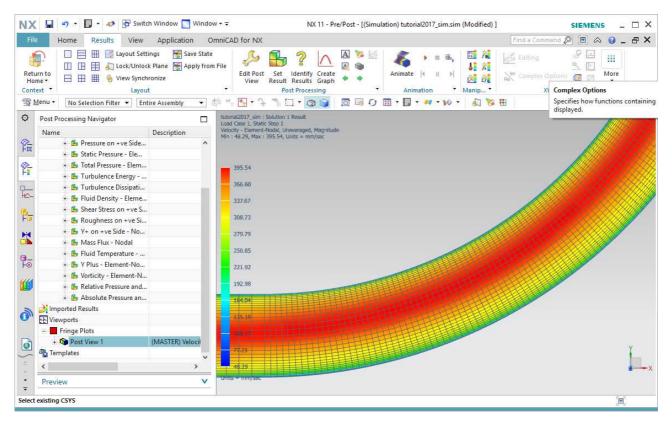
5.4 Plot the results

On the very left tool bar click the second icon to go to post-processing navigator.

*Right click on Solution**Flow > Load*

In the arborescence under flow you can access the different data field: velocity, pressure,...

To plot the velocity for instance, right click velocity > Plot



You can edit the plot by right-clicking the active Post-view>Edit. You can plot isocurves, or streamlines, etc.. You can also remove the display of the mesh by selecting, "Feature" under Edges&Faces\Primary Display\Edges.

Once you have plotted a data field, like velocity or pressure for instance, you can also create 2D graph under Post-processing toolbox.

In this tutorial we will plot the velocity profile along the pipe.

Post-processing>Create Graph

Under X Axis, Define by "Path Length"

Under Y Axis, Tick Define By Query Curve

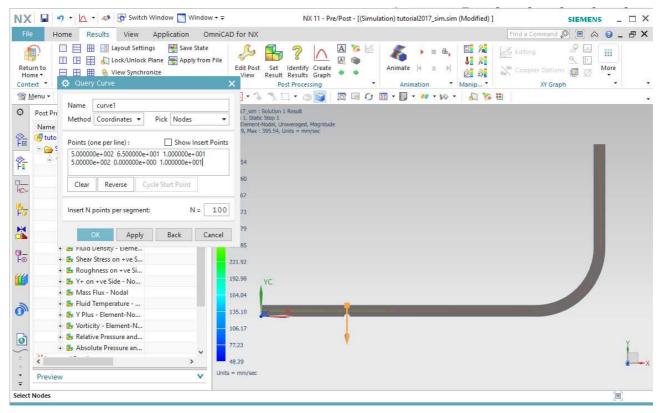
Under the "Query curve" tab, click the top right icon to create a new curve.

Give a name to the curve and select Method "coordinates"

You can then either click the two extremities of the curve on the mesh display or enter the points coordinates manually.

Here we will create a curve perpendicular to the pipe at 500mm after the entry.

Finally, enter a high number of points per segment, preferably much higher than the number of elements, 100 for instance.



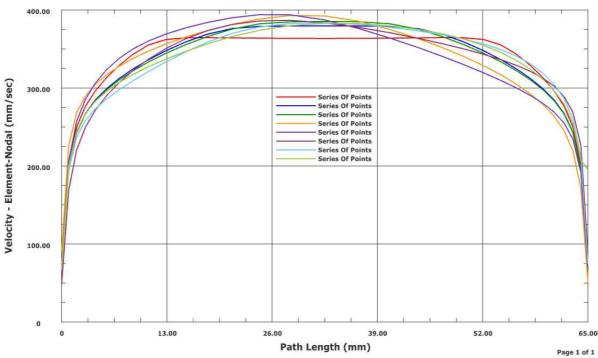
When back in the graph tab, make sure the slider for "Distance to Mesh" is at the minimum 0,0001 and select the Projection Vector to be along z-axis.

NX	1	🚯 Graph 🕠	× •=	NX 11 - Pre/Post - [(Simulation) tutorial2017_sim.sim (Modified)] SIEME	NS _ 🗆 X
File	2	Туре	nniCAD	for NX Find a Command 🔊 📵 ሪ	× 🛛 _ ₽ ×
1	2	On Path	File	A A A A A A A A A A A A A A A A A	
	irn te	Graph Title		Edit Post Set Identify Create View Result Results Graph Animate K II H AIR Complex Options II I I I I I I I I I I I I I I I I I	More
Conte		Velocity - Element-Nodal		Post Processing Animation Manip XY Graph	• •
TR N	<u>l</u> enu	X Axis	A 9 94	5 * 4 * 1 🗆 * 1 🕼 🚺 🗔 🖓 🖸 🖽 * 🖉 * 10 * 10 * 10 * 10 * 10 * 10 * 10 * 1	
¢	Po: Na	Define By Path Length	Load Ca Velocity	1017_sim : Solution 1 Result se 1, Statuti Step 1 - Demont-Hokal, Unaveraged, Magnitude	
备			Min : 4	.29, Max : 395.54, Units = mm/sec	
-		Define By Query Curve	- 39	5.54	
-		Query Curve			
P-		Curve Name curve1 🔹			
10.		×		2.67	
₽		Curve Usage Project to Element Fac 💌	30	8.73	
		Distance to Mesh Tolerance	27	9.79	
		.0001 1.0000	- 25	0.85	
₽-			22	1.92	
"		Projection Direction		2.98	
		Select Vector(1)		4.04	
0		Preview		5.10	
		Element Nodal Value Average	10	6.17	
		Error Handling	∧ 77	23	¥
-	<	No Data Ignore	- 1.7 March 13	29	a →××
* *	Pr	.	Units =	mm/sec	
Enter I	Ds	OK Apply Cancel		31 Free Points Selected	

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 a new window.



Repeat the same steps to plot the velocity profile at different length along the pipe to see how the turbulent flow profile goes from flat at the 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.



Velocity - Element-Nodal

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/11/nx_help/train ing/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" to exit the result analysis and go back to the simulation navigator.

In the arborescence, 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 (yourproject.prt), select the "Part Navigator" (3rd icon in the left vertical toolbar).

Now you can edit your sketch by double-clicking it. Change the radius from 400 to 70mm.

Click finish the 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 the top left "Activate the Simulation" 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.

Image: Sevent settings Image: Sevent settingsevent setting Image: Sevent setting	X					SIEMENS		_	Y
Simulation Navigator Intonia2017, sim : Solution 1 Realt tool sock in the series of the series	etur Hom Inte	Layut Sin Layut Sin L	ettings ock Plane ochronize	Save State Apply from	File File Complex Create View Result Scraph Post Processing View Complex Create Post Processing View Complex Create View Com		More		
Name Color Status Witch: Displayed & * @ tutorial2017_sim.sim Displayed & * @ tutorial2017_fem.fem ************************************	M	enu 👻 No Selection Filter 👻 🛛	Entire Asse	mbly 🔻	8 ° ° ⊡ + 4 ° ° ⊡ + ⊘ ⊘ □ ○ □ • □ • ○ • 				8
Name Color Status Image: Color Status Image: Color Displayed & Image: Color Displayed & Image: Color Status Image: Color Displayed & Image: Color Status Image: Constraint Container Status Image: Constraint Status Status	6	Simulation Navigator							
• CSVS	H	🐠 tutorial2017_sim.sim			Min : 0.26, Max : 720.15, Units = mm/sec				
TN Fields 660.16 Modeling Objects (Filtered) 600.17 Filter Constraint Container 540.18 Constraint Container 480.19 Solver Sets 420.20 Solver Sets 420.21 Solver Sets 180.23 180.23 120.24 120.25 0.26		+ 🗹 📴 CSYS			720.15				
+ □ + ○ Simulation Object Cont 540.18 + □ + ▷ Load Container 480.19 - ★ Load Container 420.20 - ♦ ○ Simulation Objects 360.21 - ↑ ▷ Simulation Objects 300.21 - ↑ ▷ Constraint Set 300.21 - ↑ ▷ Results 180.23 - ↓ ≥ Constraint Set 120.24 - ▷ ≥ Constraint Set 10.25 - ▷ ≥ Constraint Set 10.26		[]]F[x] Fields							
Provide	•	+ 🖸 🔑 Simulation Object Cont			540.18				
→ 300/ef sets 360.21 → 1 → 1 → 1 → 1 → 1 → 1 → 1 → 1 → 1 → 1		Load Container							
□ 1/2 Constraint Set □ 1/2 Loads + □ Results 180.23 120.24 50.25 0.26		- 🤕 Solution 1		Active					
* Results 186.23 120.24 100.25 0.26		Constraint Set							
50.25 0.26	1								
0.26									
					0.26				
								1	Ĩ

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" button to get the new values.

Solution 🔺	Quantity	Component	Operation	Selection Type	Value	Units
Solution 1	Static Pressure	##11Scalar	Minimum	Entire Model	Pmin=-281.58	Pa(N/m^2)
Solution 1	Static Pressure	##11Scalar	Maximum	Entire Model	Pmax=81.572	Pa(N/m^2)
Solution 1	Static Pressure	##11Scalar	Mean Average	Geometry	Pinlet=75.134	Pa(N/m^2)
Solution 1	Velocity	Magnitude	Minimum	Entire Model	Vmin=0.0011318	m/sec
Solution 1	Velocity	Magnitude	Maximum	Entire Model	Vax=0.72136	m/sec

	Large radius (400mm)	Short radius (70mm)
Inlet average pressure	35,9 Pa	75,1 Pa
Maximum velocity	0,395 m/s	0,721 m/s
Maximum pressure	38,0 Pa	81,57 Pa
Minimum pressure	-3,0 Pa	-284,6 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.

7 Extra Resources

NX 11 help menu :

https://docs.plm.automation.siemens.com/tdoc/nx/11/nx_help/#uid:index

Many tutorials are available here :

https://docs.plm.automation.siemens.com/data_services/resources/nx/11/nx_help//trai ning/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/11/nx_help/com mon/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