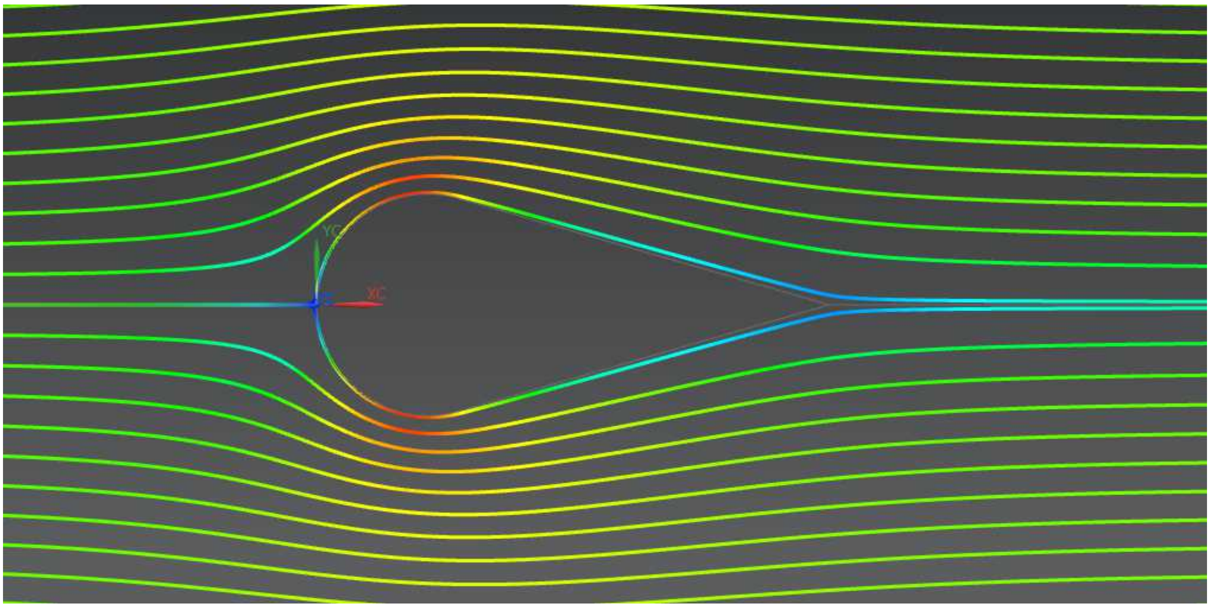
How to set-up and run a 2D Flow Simulation in Simcenter NX11



[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 :

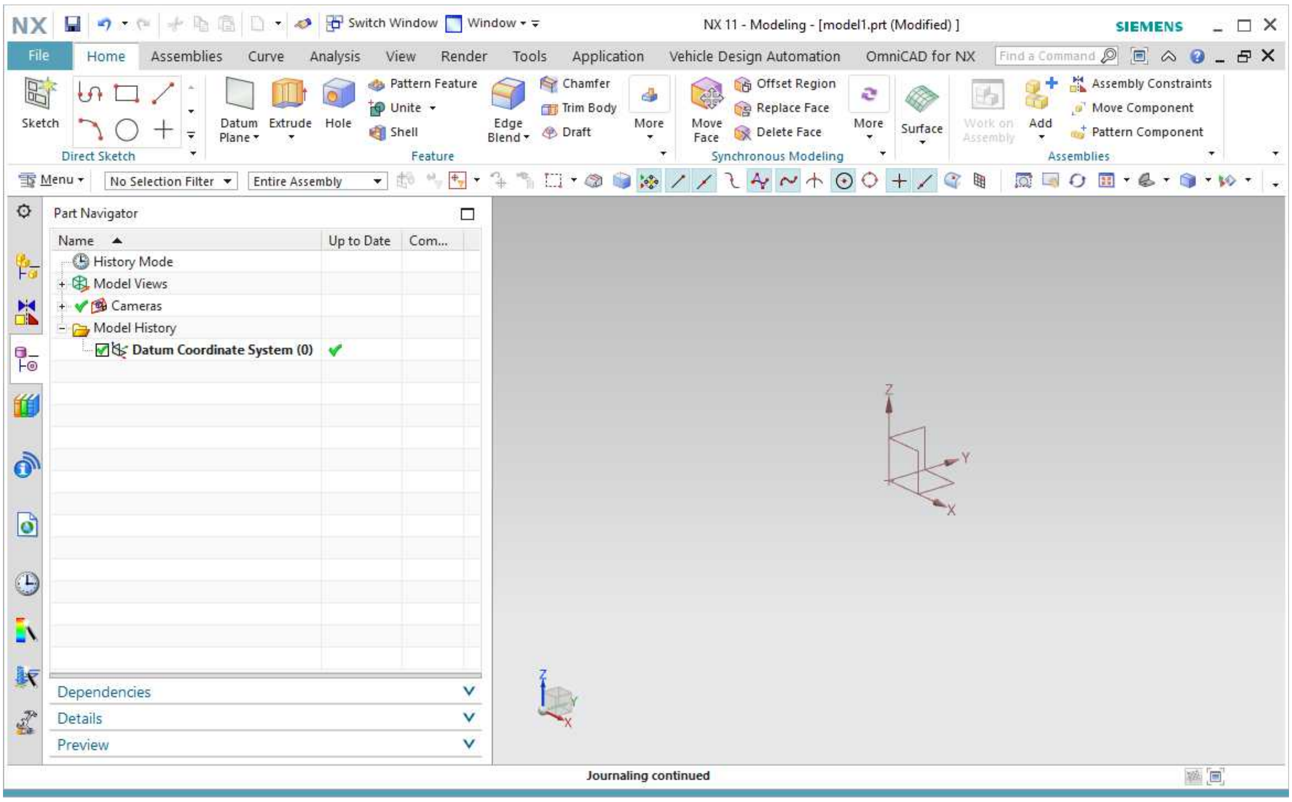
https://docs.plm.automation.siemens.com/tdoc/nx/11/nx_help#uid:xid1128419:index_advanced:id1245911:id629501

1 Create the fluid domain

1.1 Create an empty model

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

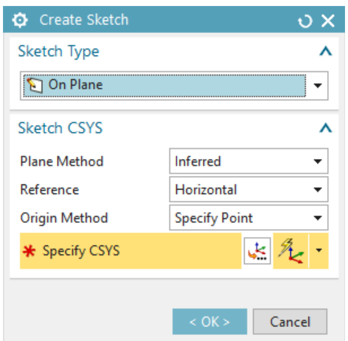
You shall have this view and you are ready to create your part.



1.2 Create a 2D sketch

Click "Sketch" in the upper left corner

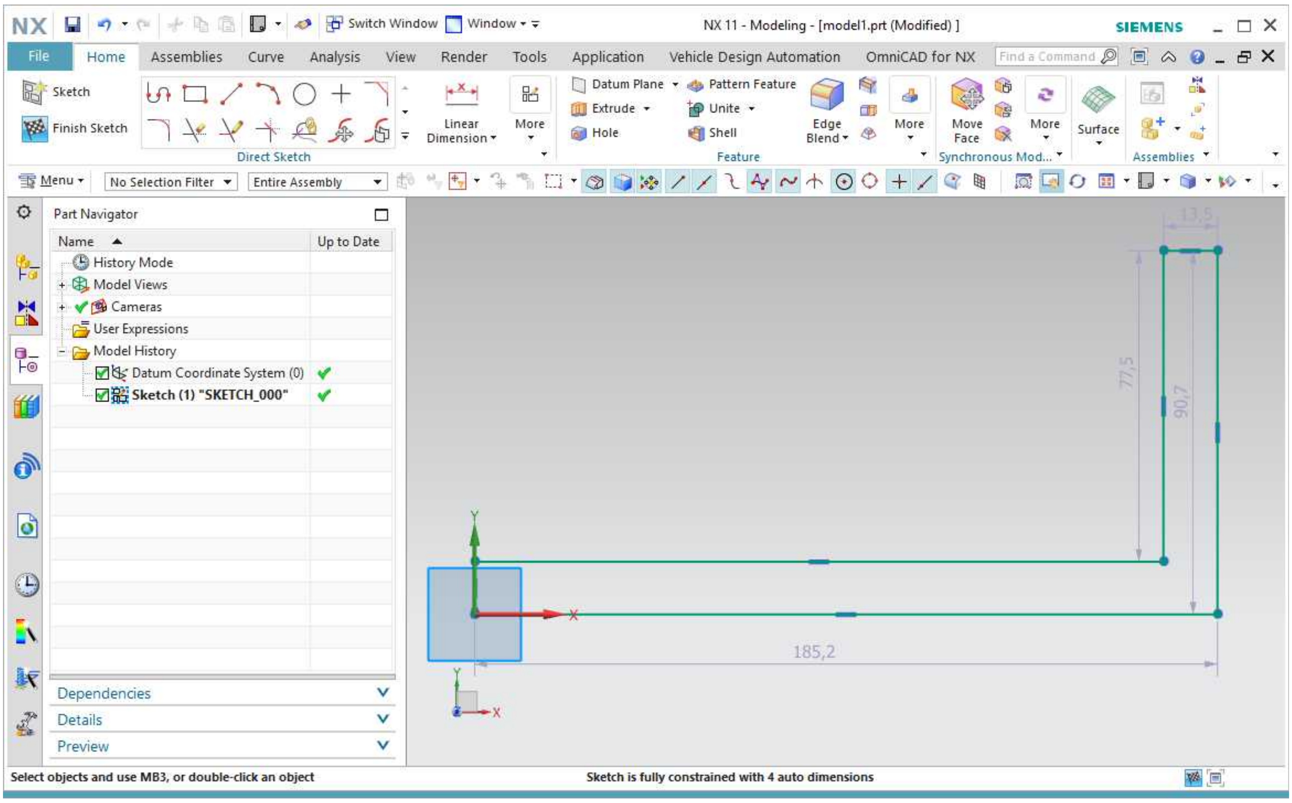
You can keep the default coordinate axis>OK



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.

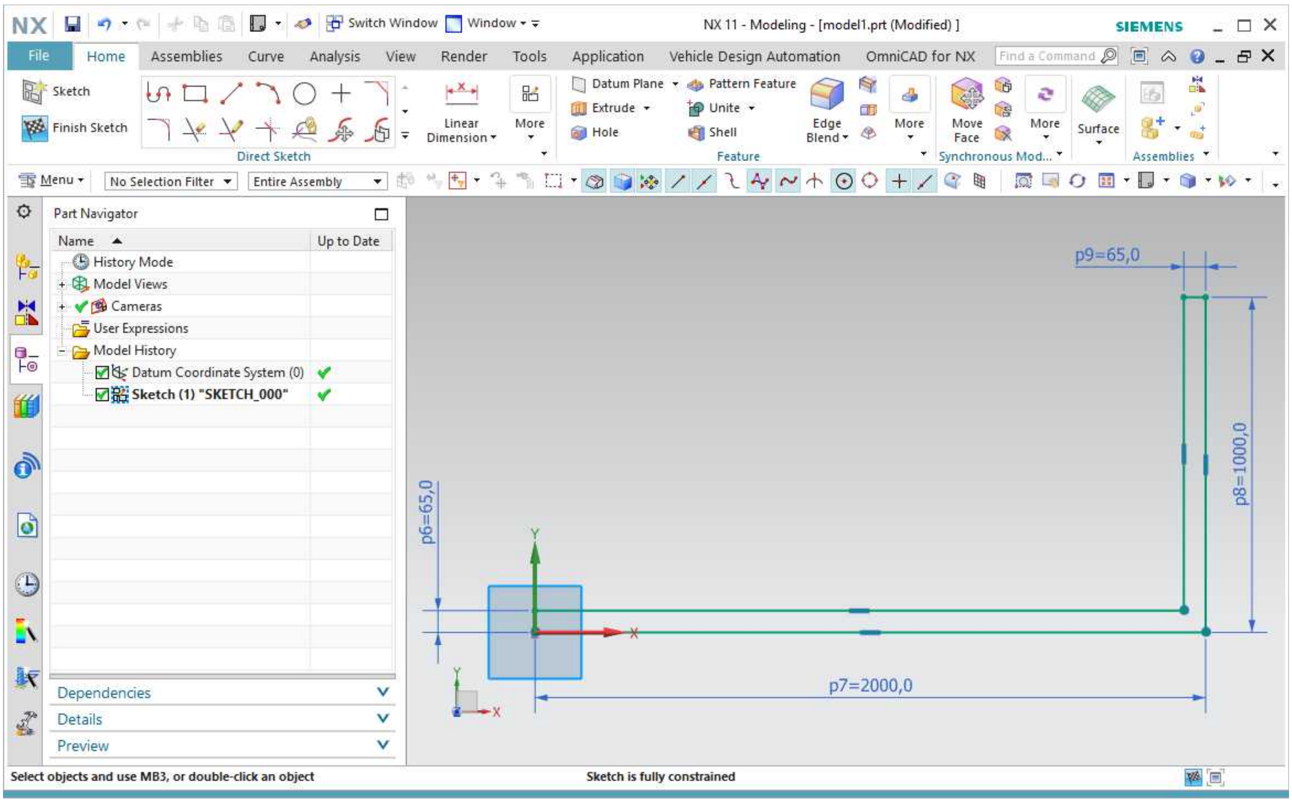


You will see that grey dimensions appear automatically in such a way to fully define your sketch. These are auto-dimensions. You can double-click some and adjust the value to make them constraints, they will appear in dark blue color. You can add constraints by selecting a segment and then choosing “horizontal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

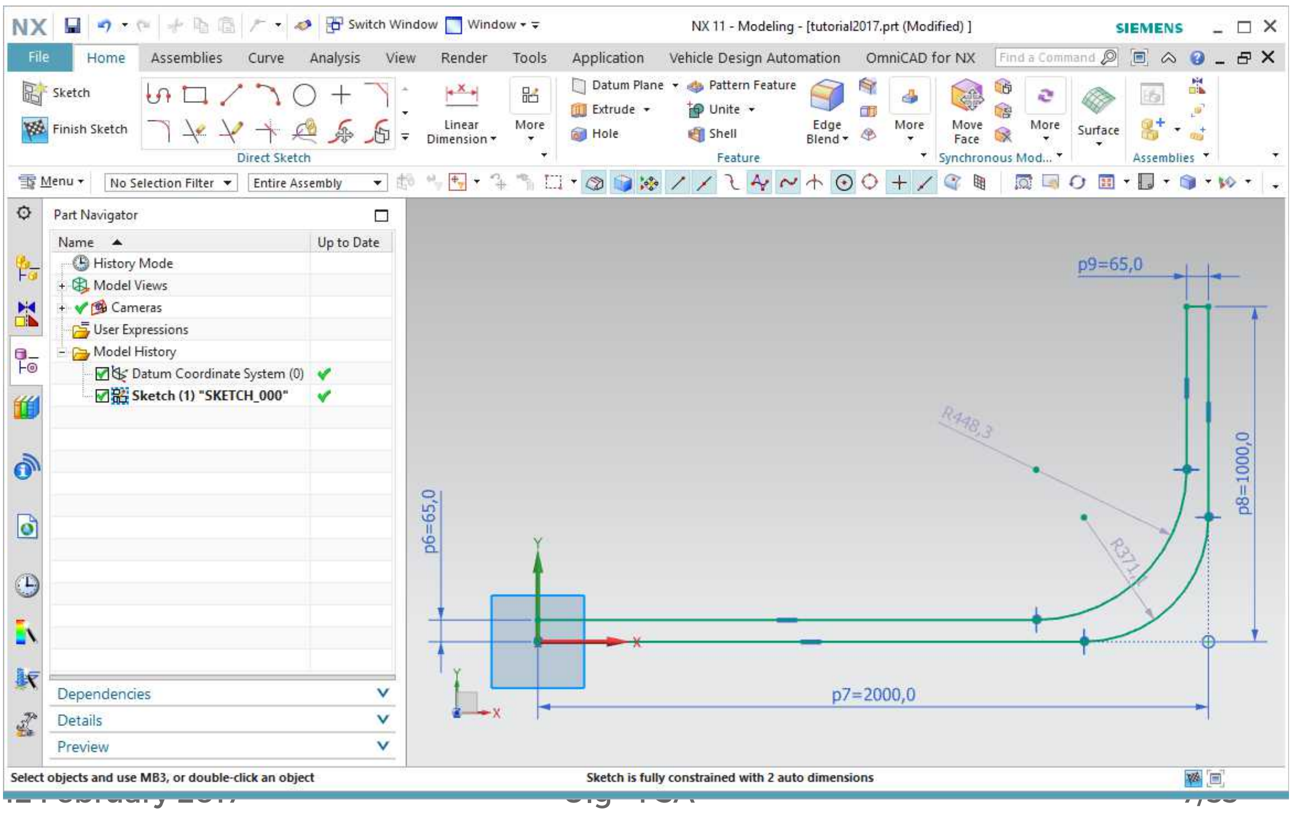
For instance, if you want a segment to be symmetrically located with respect to the coordinate axis, you select first the mid-point of the segment (with left mouse click), then the x-axis arrow (no need to hold down CTRL or SHIFT key) and then a suggestion of constraint automatically appears “point on 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.

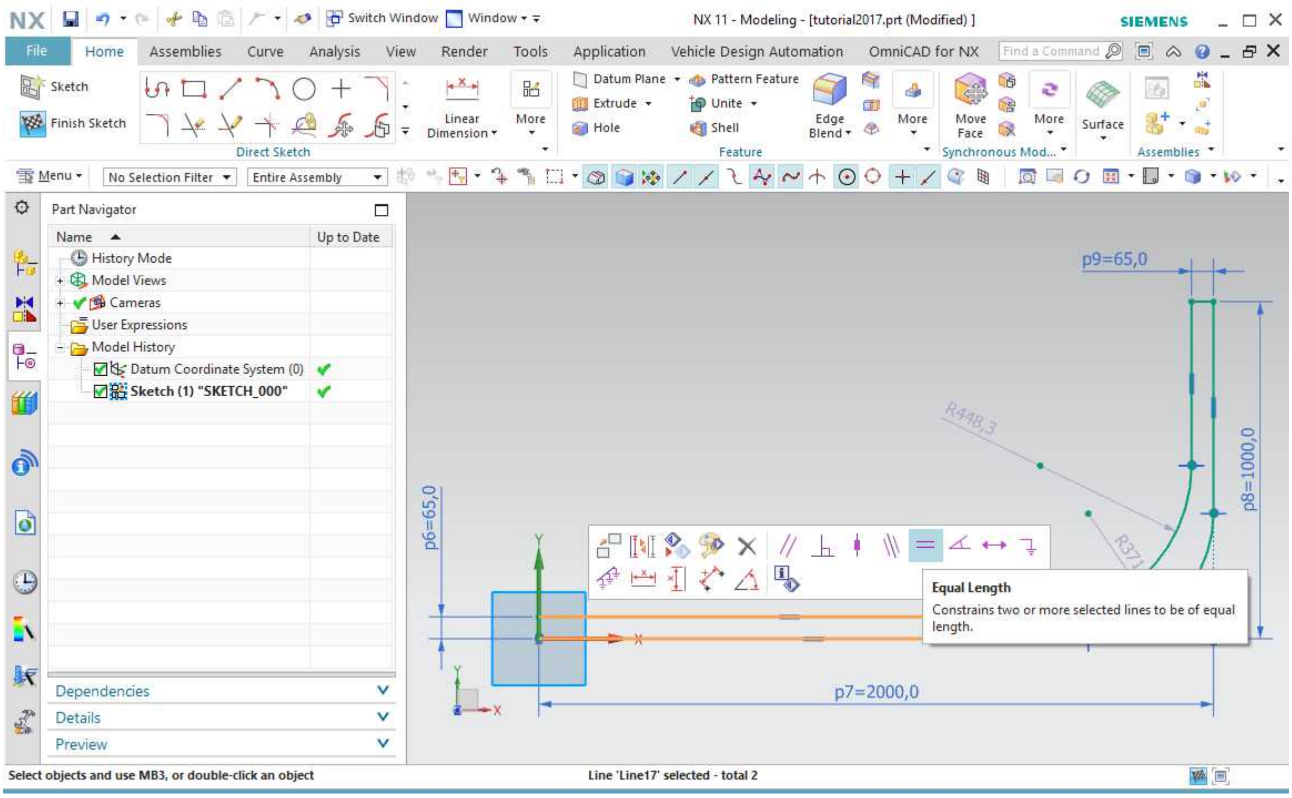


Ideally when you are done with your sketch as here-above, it shouldn't contain any grey dimensions (auto dimensions) but only dark blue constraints (px=...) set by you. This means the sketch is fully constrained. The constraint status also appear in the status bar (bottom of the window).

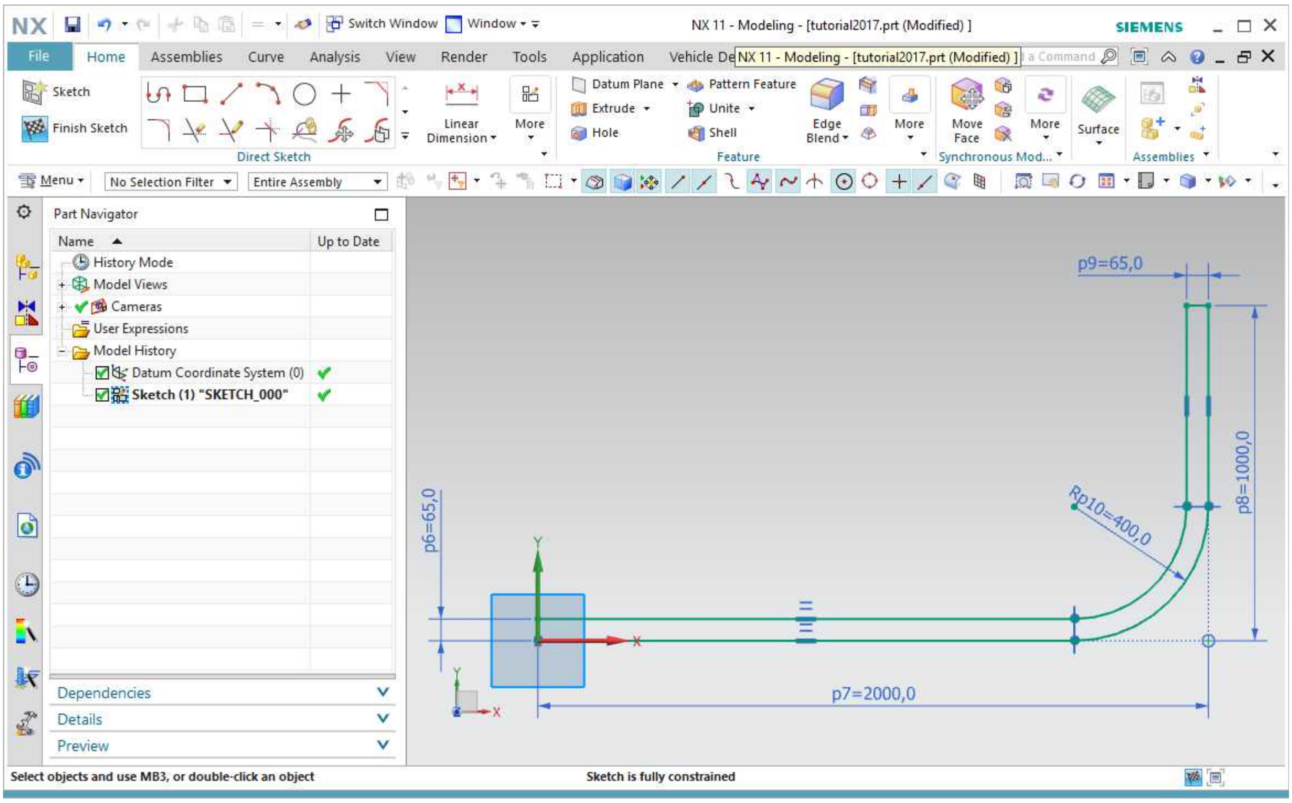
The next step is to make a round corner. Select the “filet” tool under “Direct Sketch” and then successively the two inner legs of the corner. Repeat with the two outer legs. You should have the following sketch with two additional degree of freedom, being the two arc radius.



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

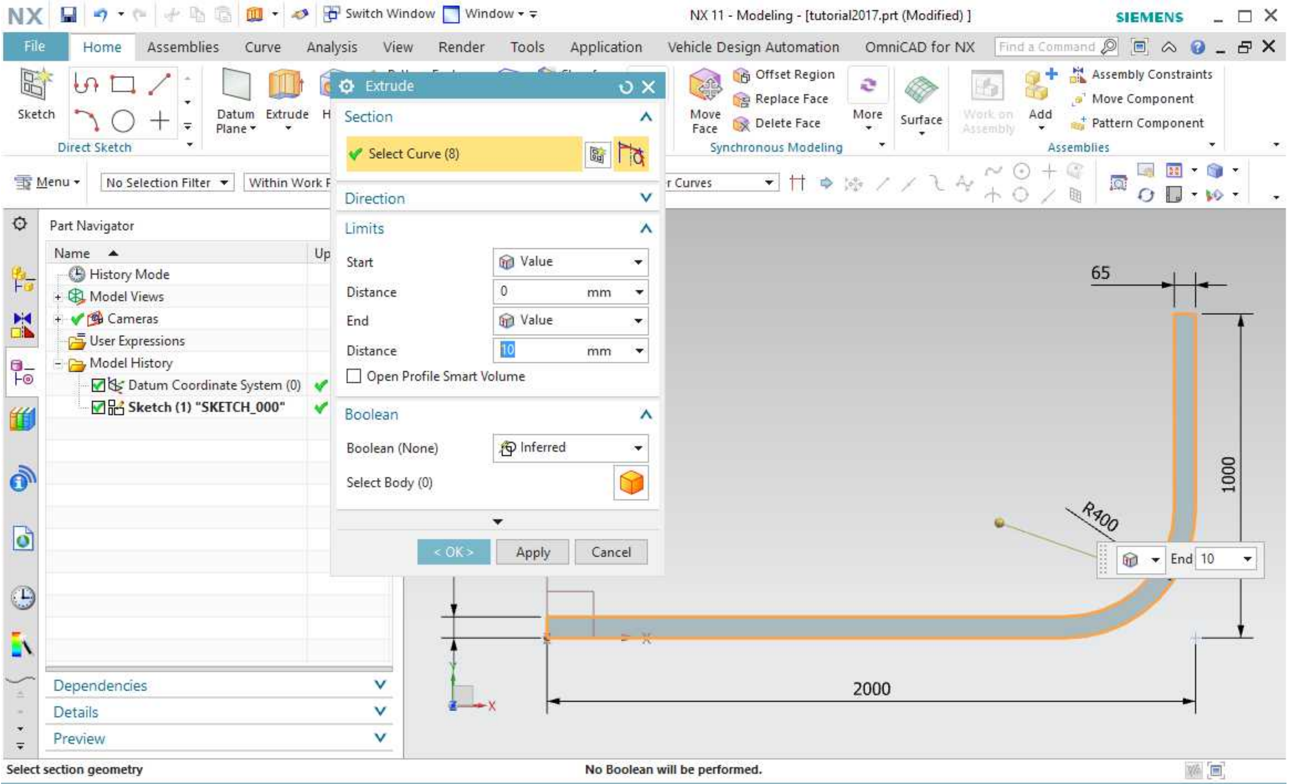
- Clicking the central mouse button and dragging will rotate the view.
- Clicking the central mouse button, then the right mouse button and dragging will move the view. You can do the same more easily by holding down the SHIFT key and clicking only the central mouse button and dragging.
- Rolling the 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

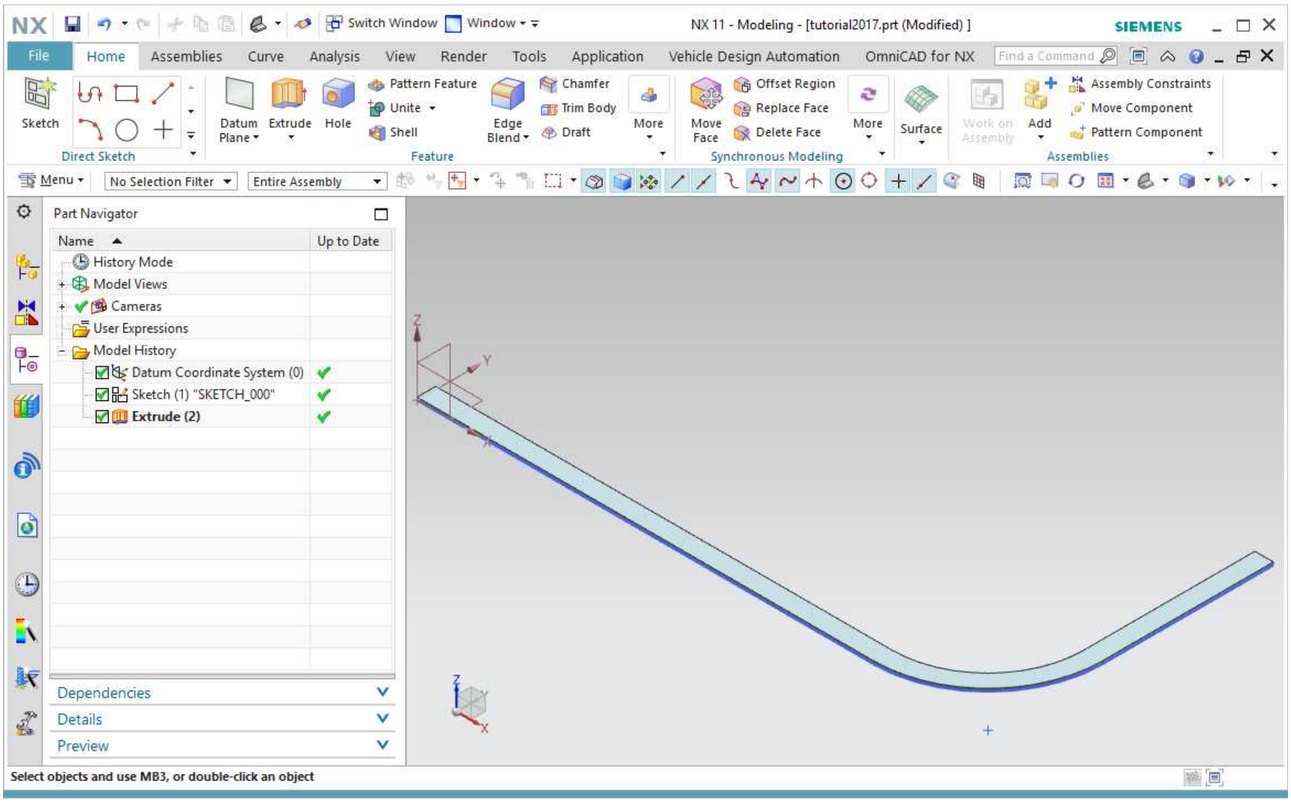
When you are done with the sketch, *Click Finish Sketch in the upper left*

1.5 Extrude the part

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



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

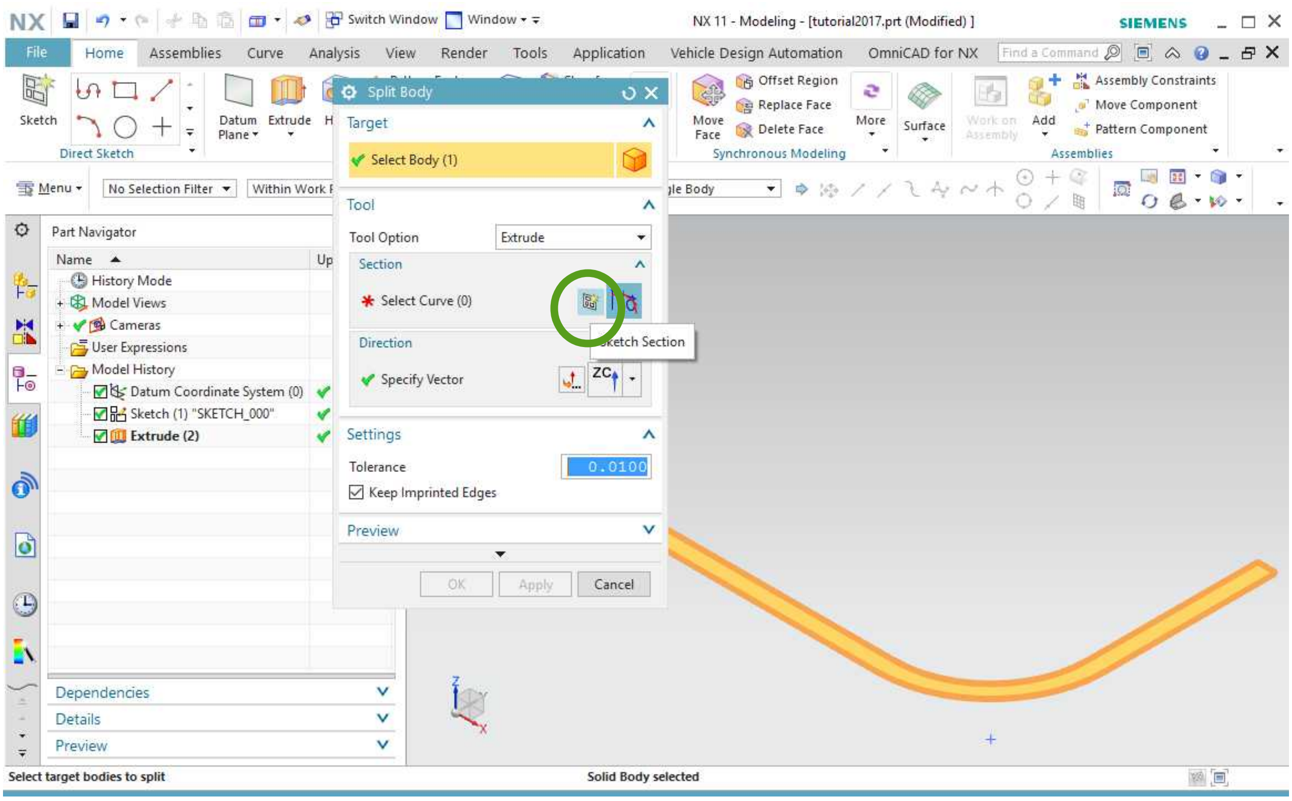


1.6 Split the body to prepare the meshing

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



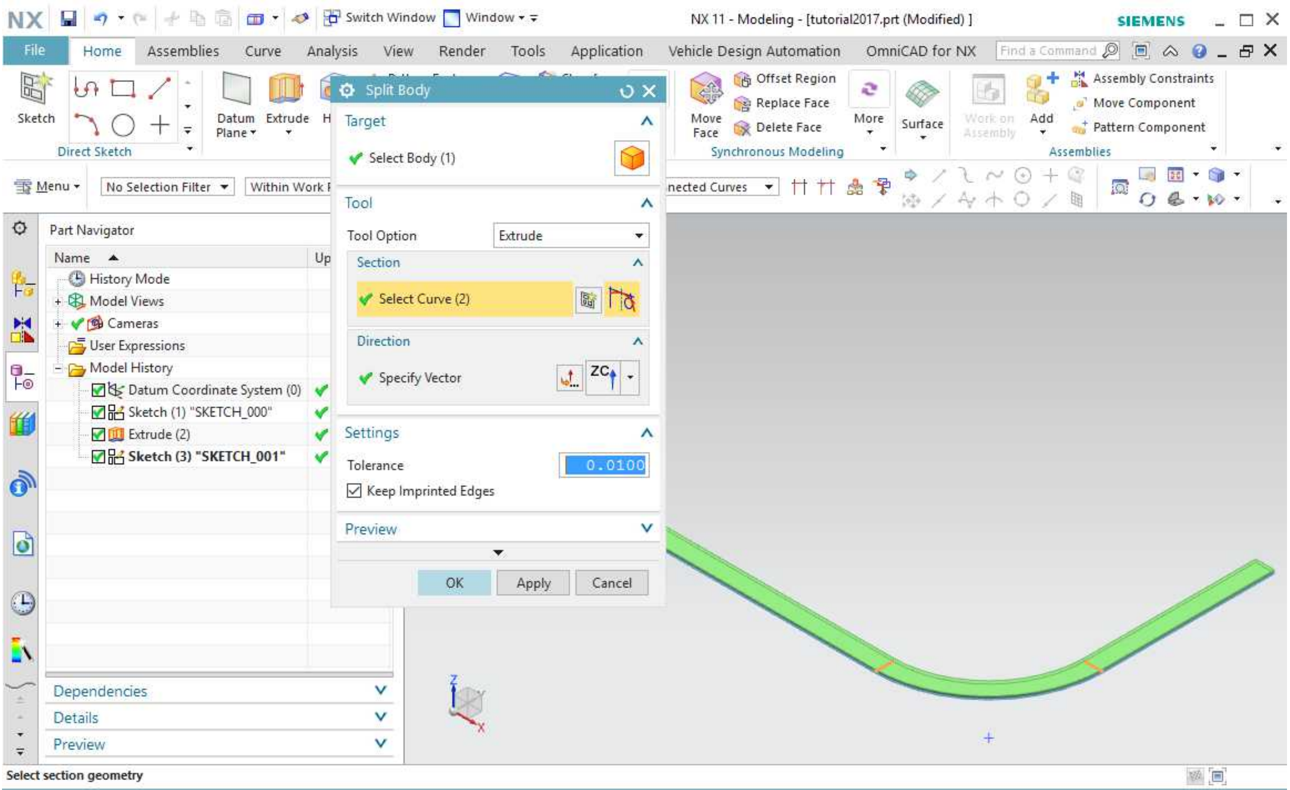
Just click OK on the next dialog box to select the same system coordinate (CSYS) you already used.

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.



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

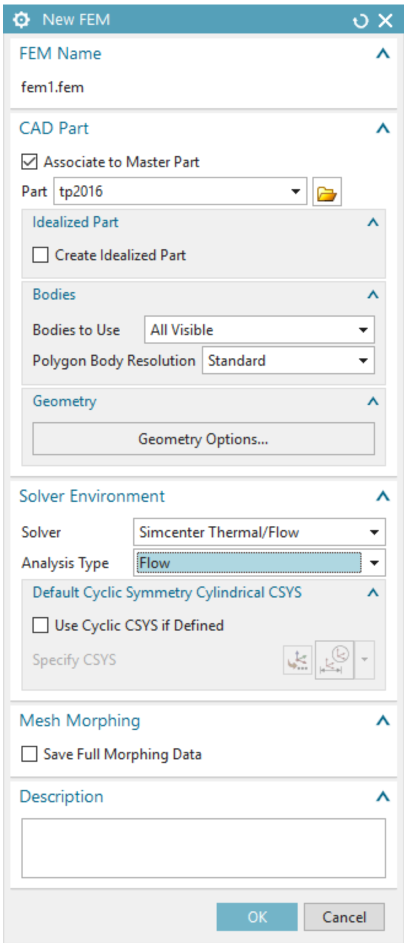
Sketch video examples can be found here :

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".

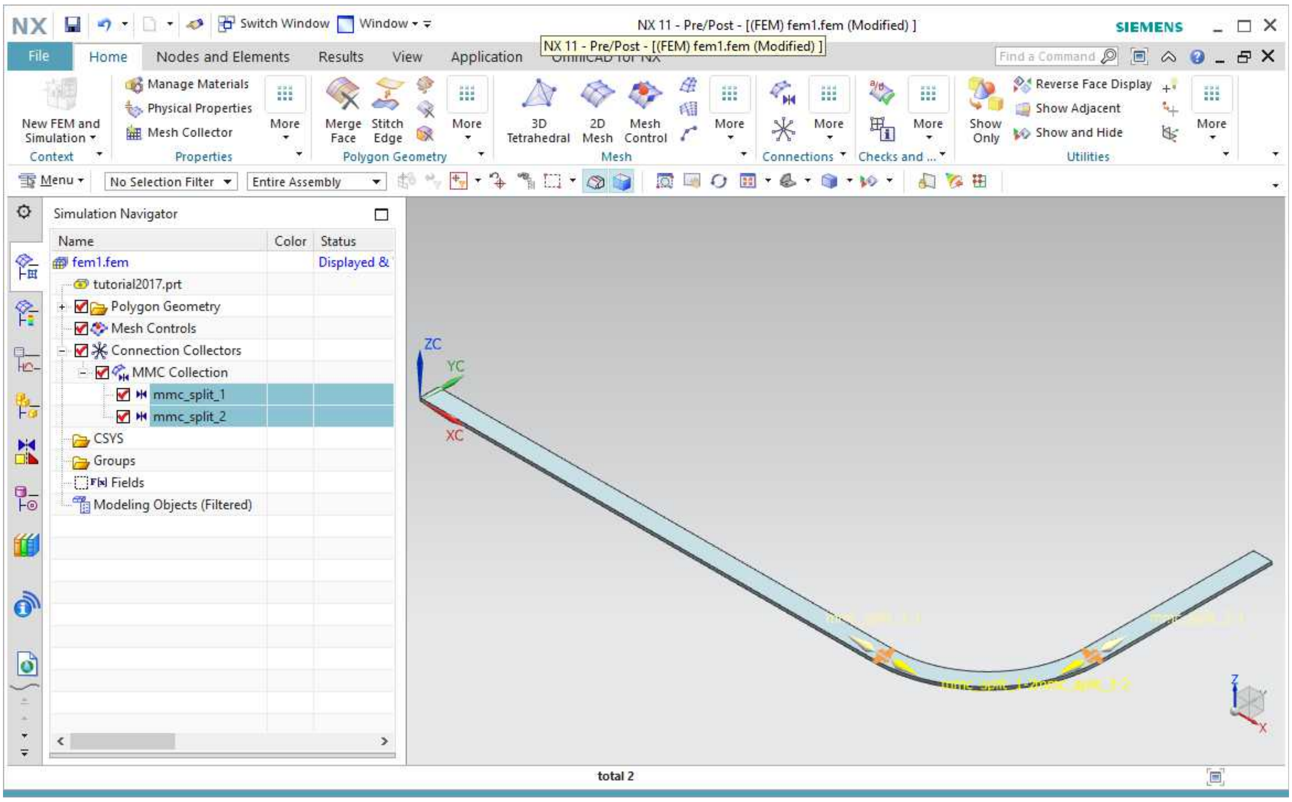
This option is used when working in team on the same part or if the part has to be simplified for the simulation but the original design has to kept unaltered.

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

Click OK.

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.

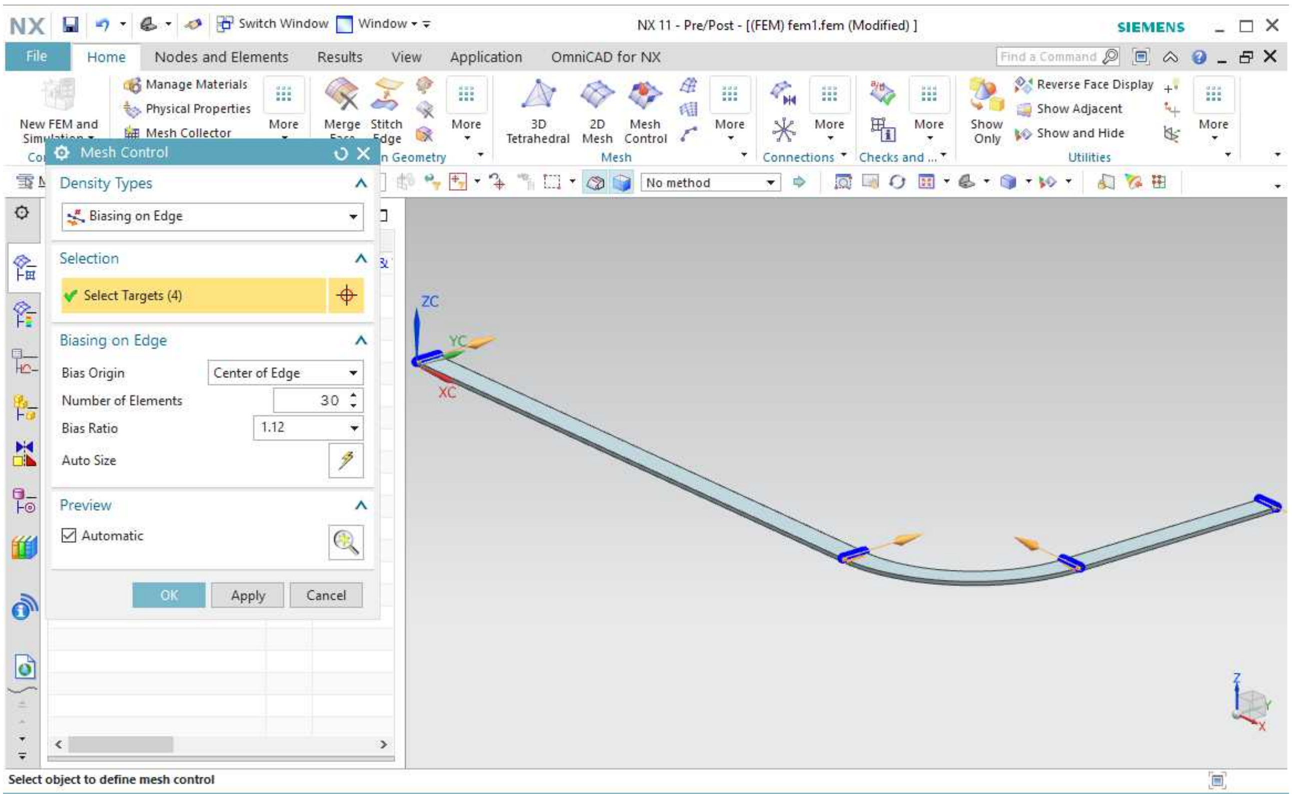
TIP : You can change the colour of your polygons by double clicking the small color box next to it in the arborescence and rename them to make identification easier later on.

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.

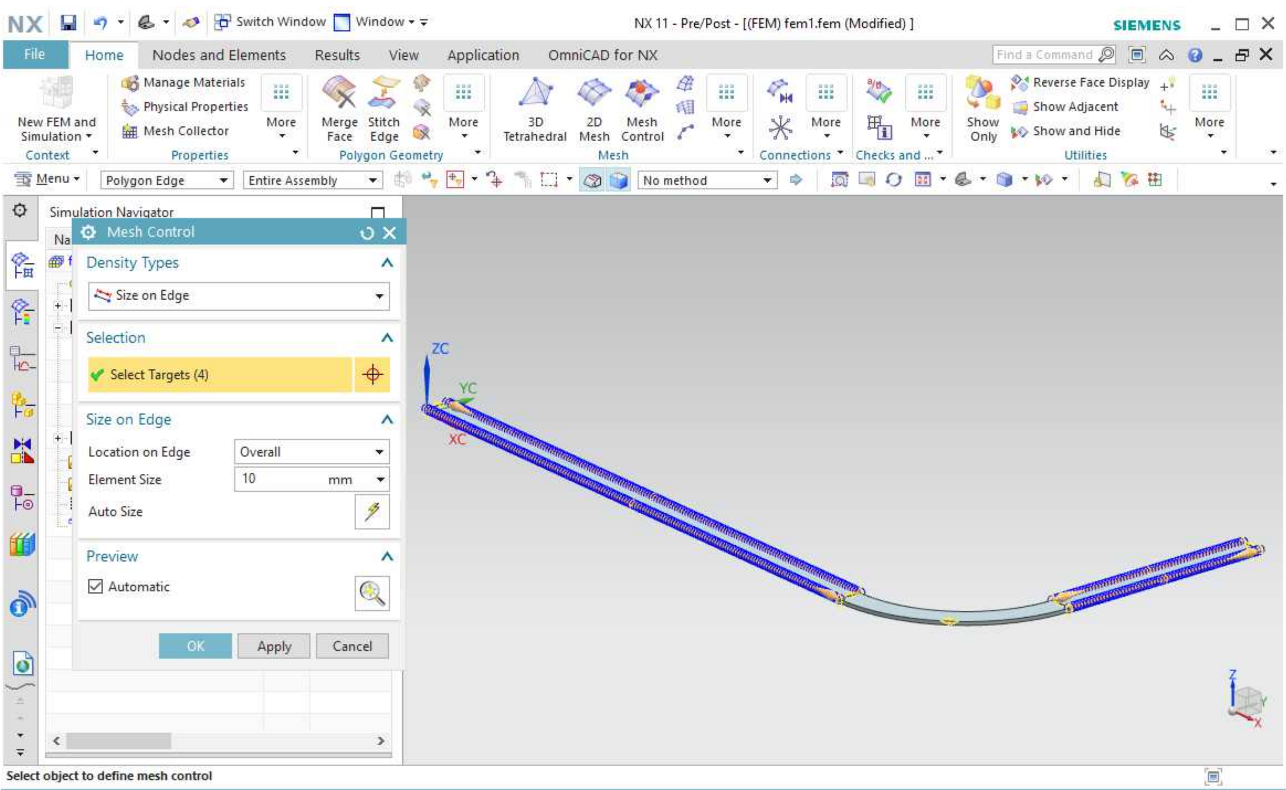
You can set a constraint on multiple edges at a time. For the biasing on edge (especially useful for boundary layer), you have to select the orientation (start of start, end of edge or middle of edge) and it varies from edges to edges depending on the edge natural direction.

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.

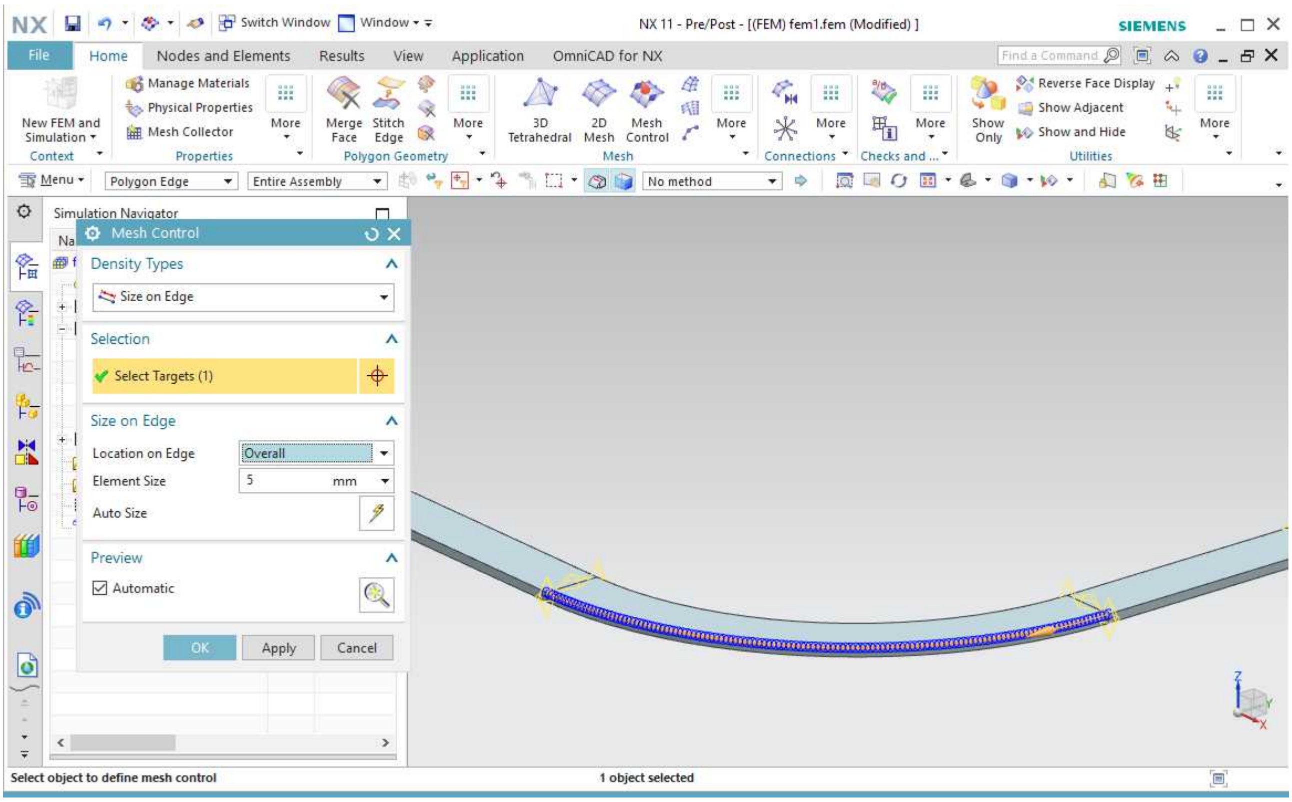


Remark : A matching edge between two polygons should be specified once only.

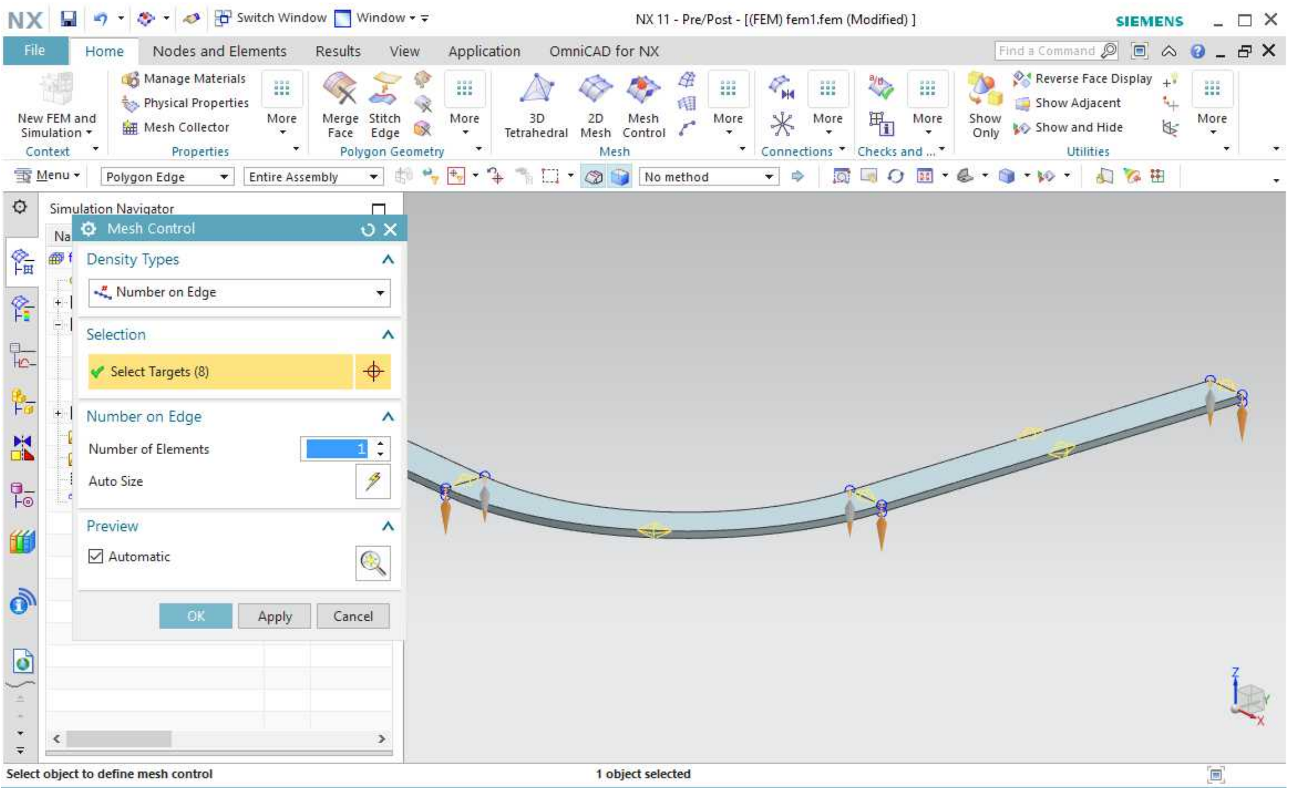
Next, you force the elements size to be 10mm on the entry and exit lateral sections. You shall put all those constraints in the same plane, here the upper plane.



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.



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

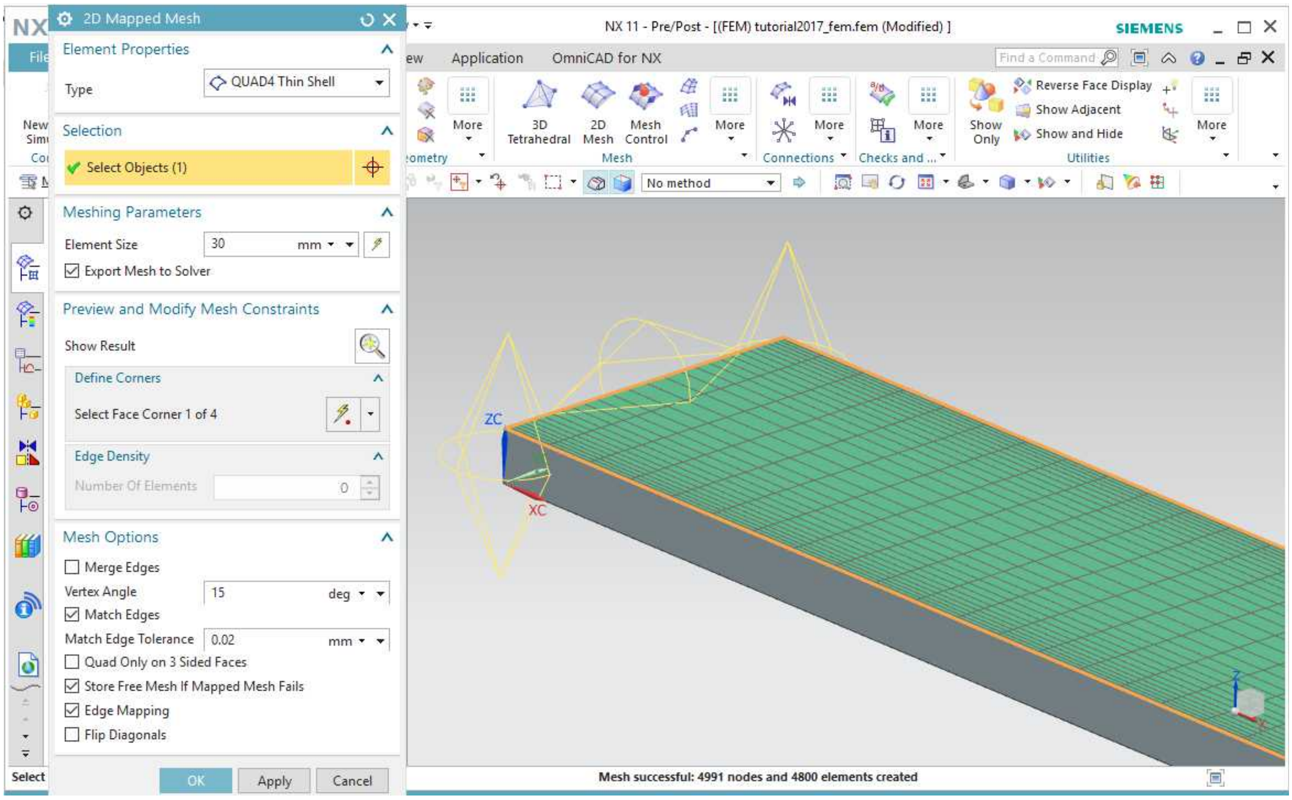
You can now mesh the face by using 2D mapped mesh. Off course you should mesh the plane on which you defined the mesh constraints. From experience, it is recommended to map the face one by one.

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 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 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.

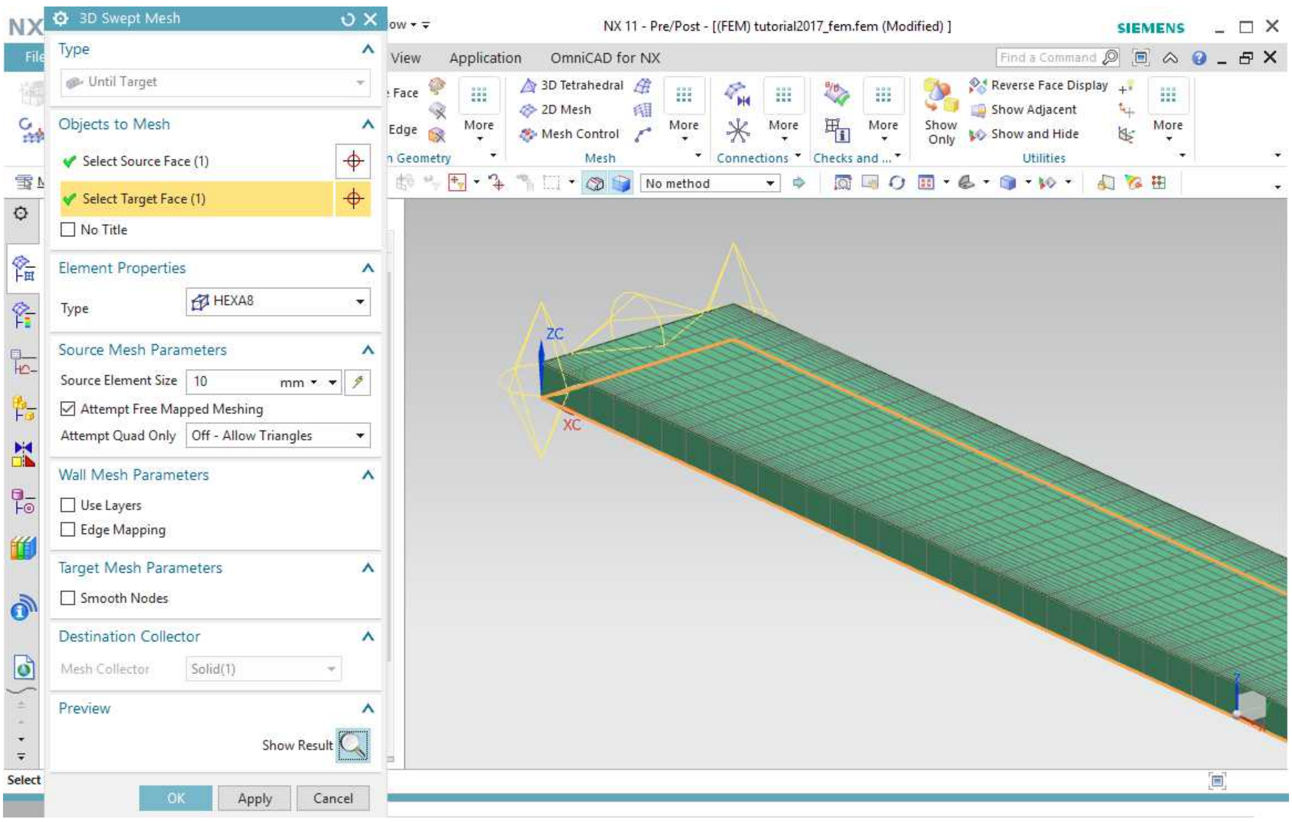
2.6 Create 3D swept mesh

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.

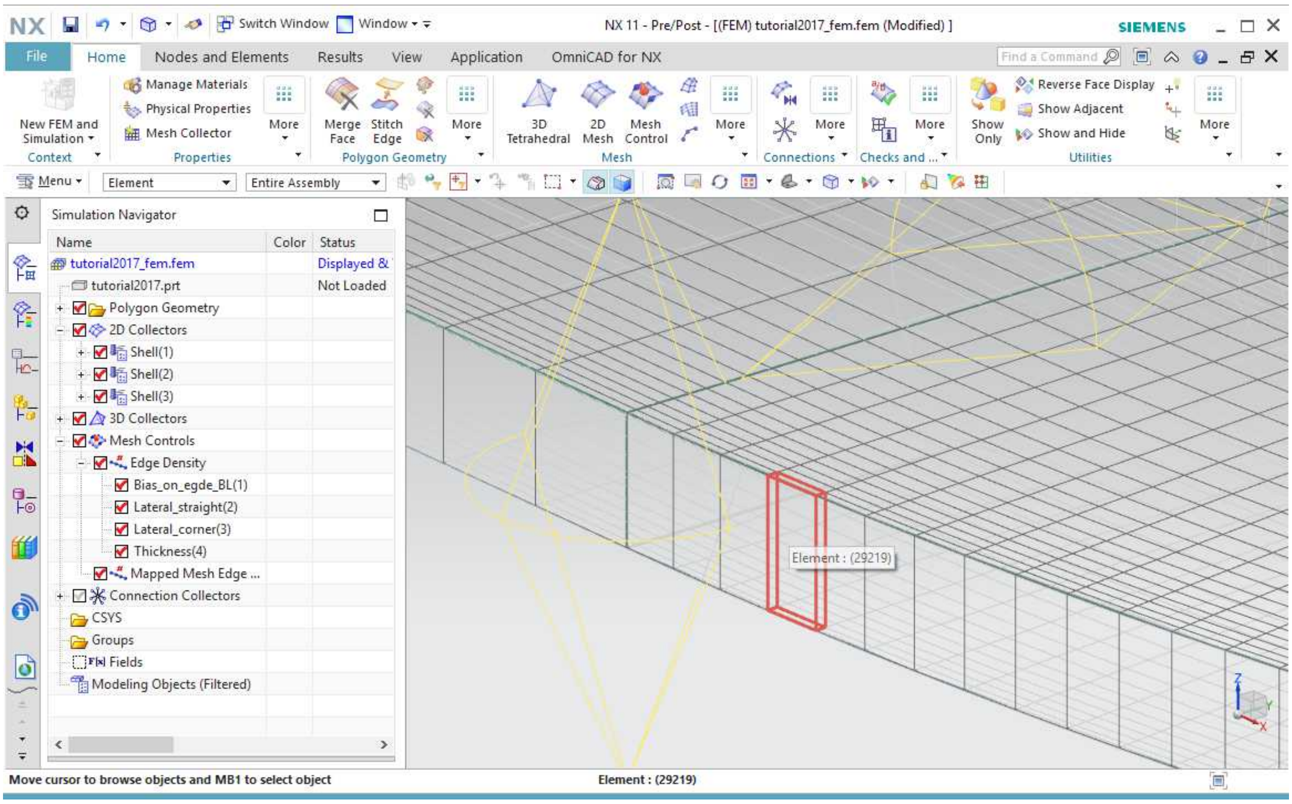
Here again, the “Source Element Size” parameter should have no effect if you correctly set a constraint of 1 single element on the edges along thickness.



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.

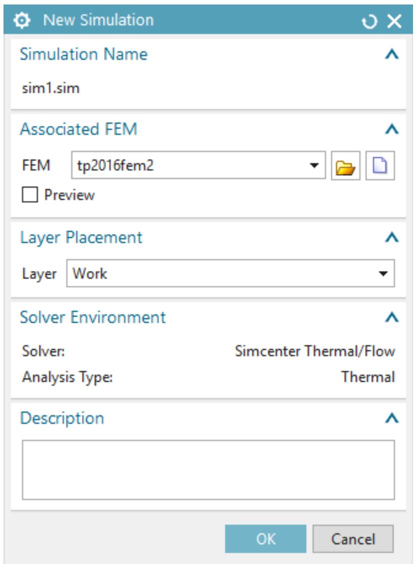
3 Specify material properties and set the simulation constraints

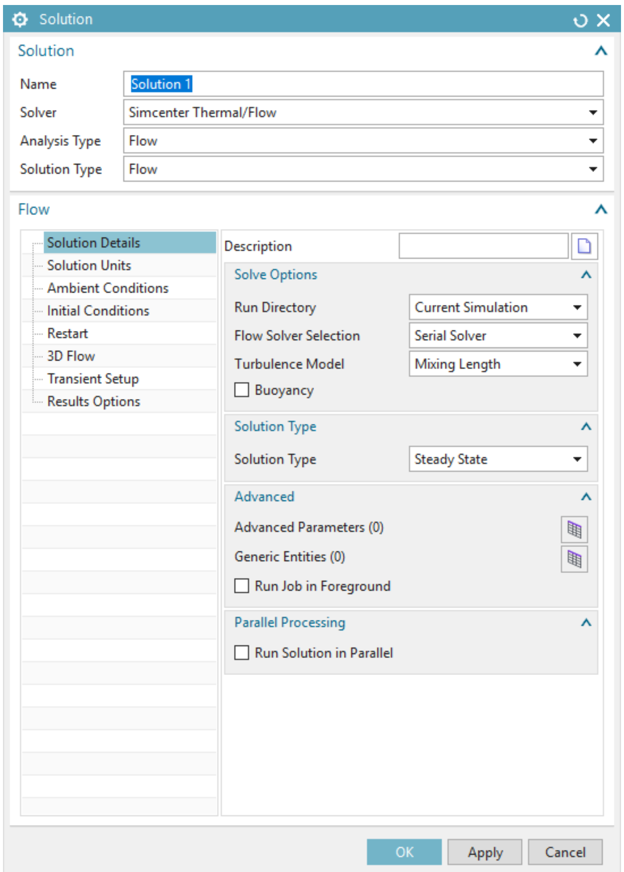
3.1 Create a sim file

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.

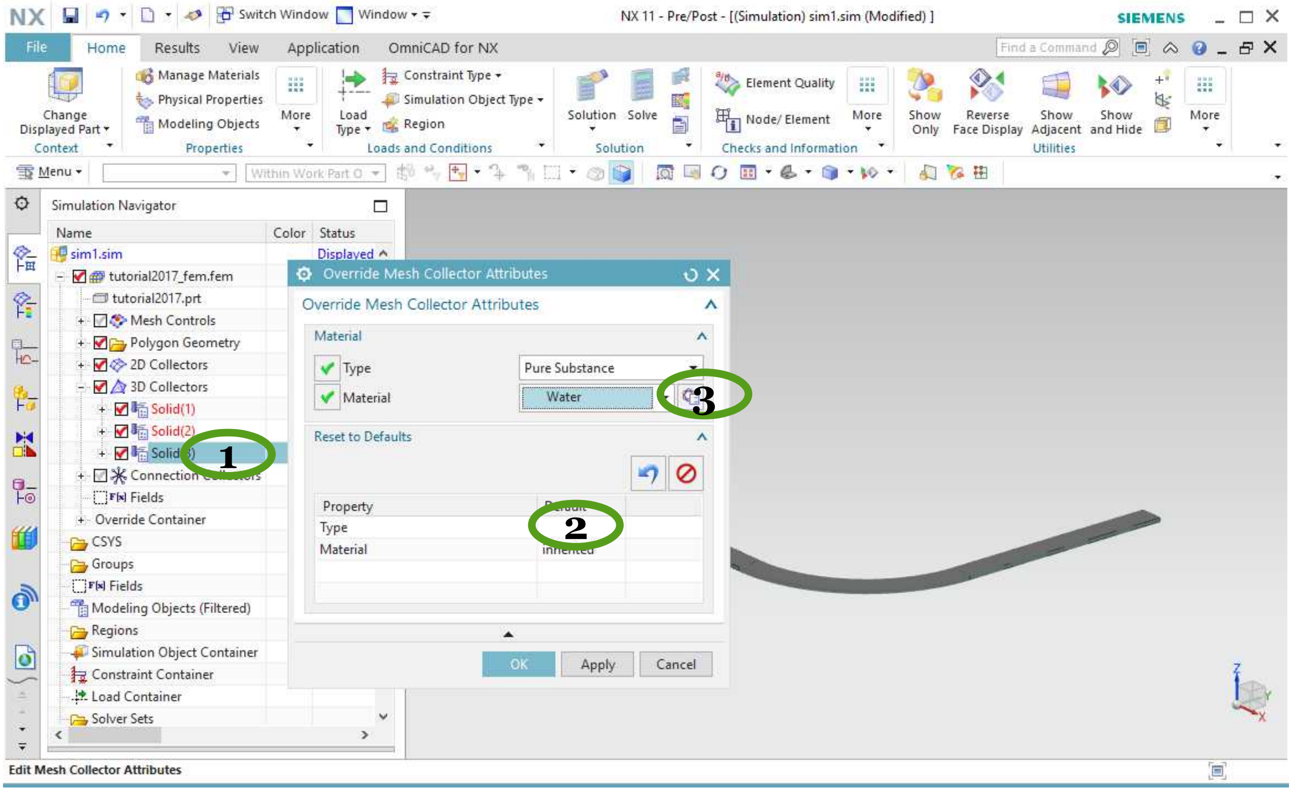




After changing the Analysis Type to Flow, Click OK, as it will still be possible to change all these solution attributes before running the simulation.

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.



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)*

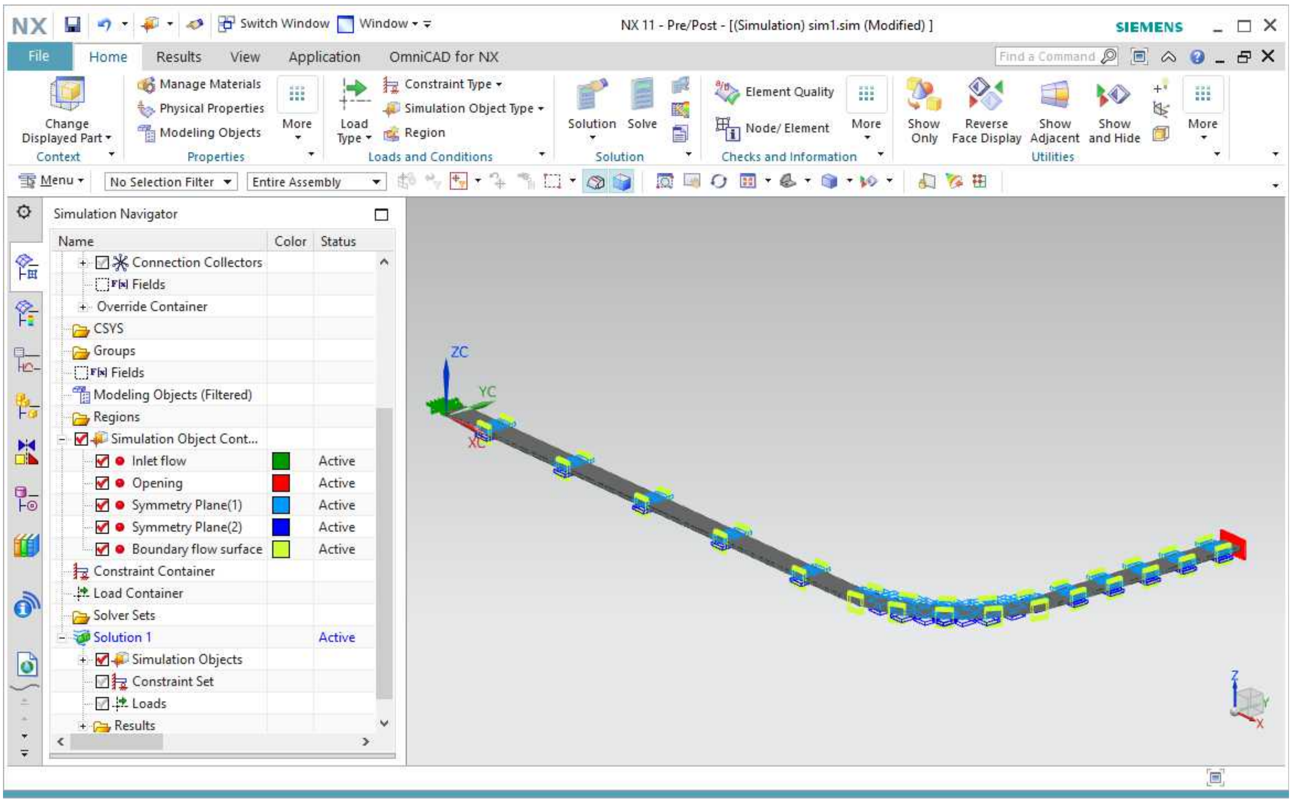
In this tutorial, you will set 0,34m/s as input velocity at the extremity of long leg of the pipe and ambient pressure at the extremity of the short leg.

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.

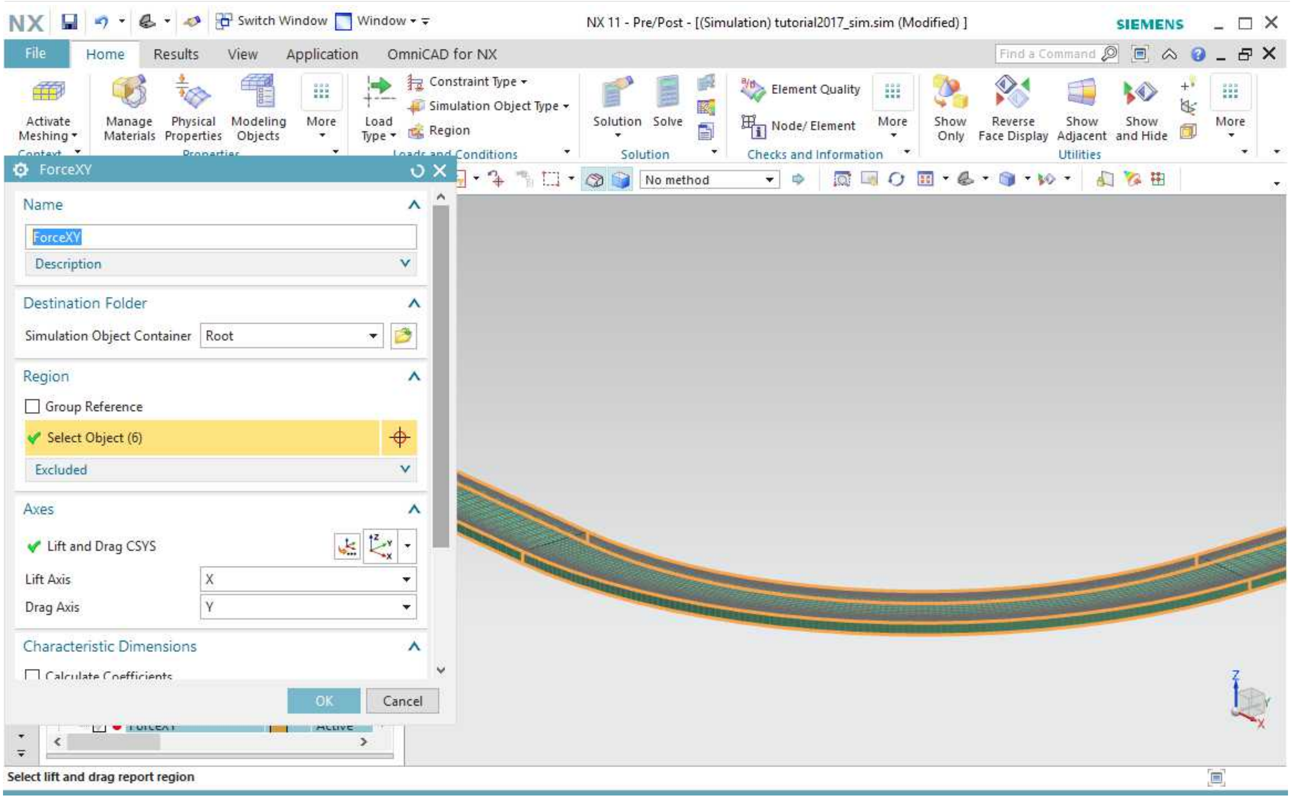
In this tutorial, this is not necessary.

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.



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.

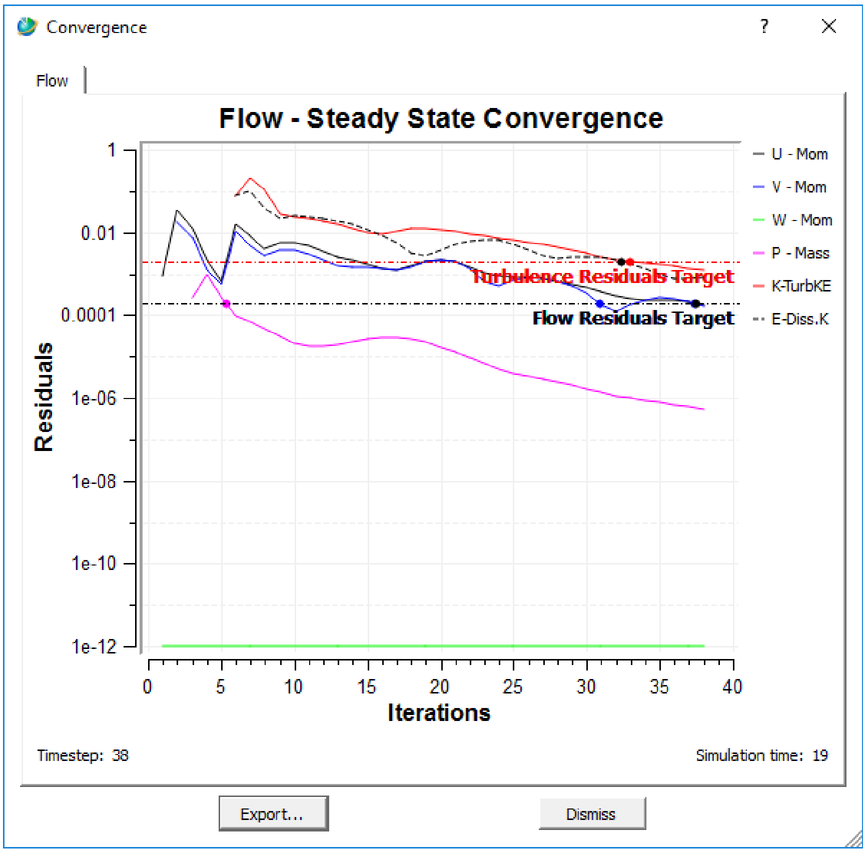
You can also change the Maximum residuals. In order to study the convergence of the solution based on the mesh refinement, you could put a smaller “Maximum Residuals”.

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.

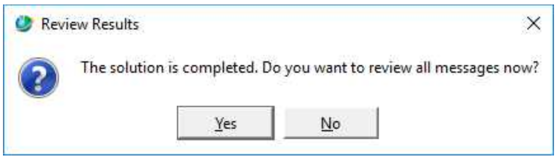


Remark : As the simulation is purely bidimensional, the residual for W (velocity along z-axis) is constant at 10^{-12}. This is a quick check for any mistake that could make the flow not purely bidimensional.

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

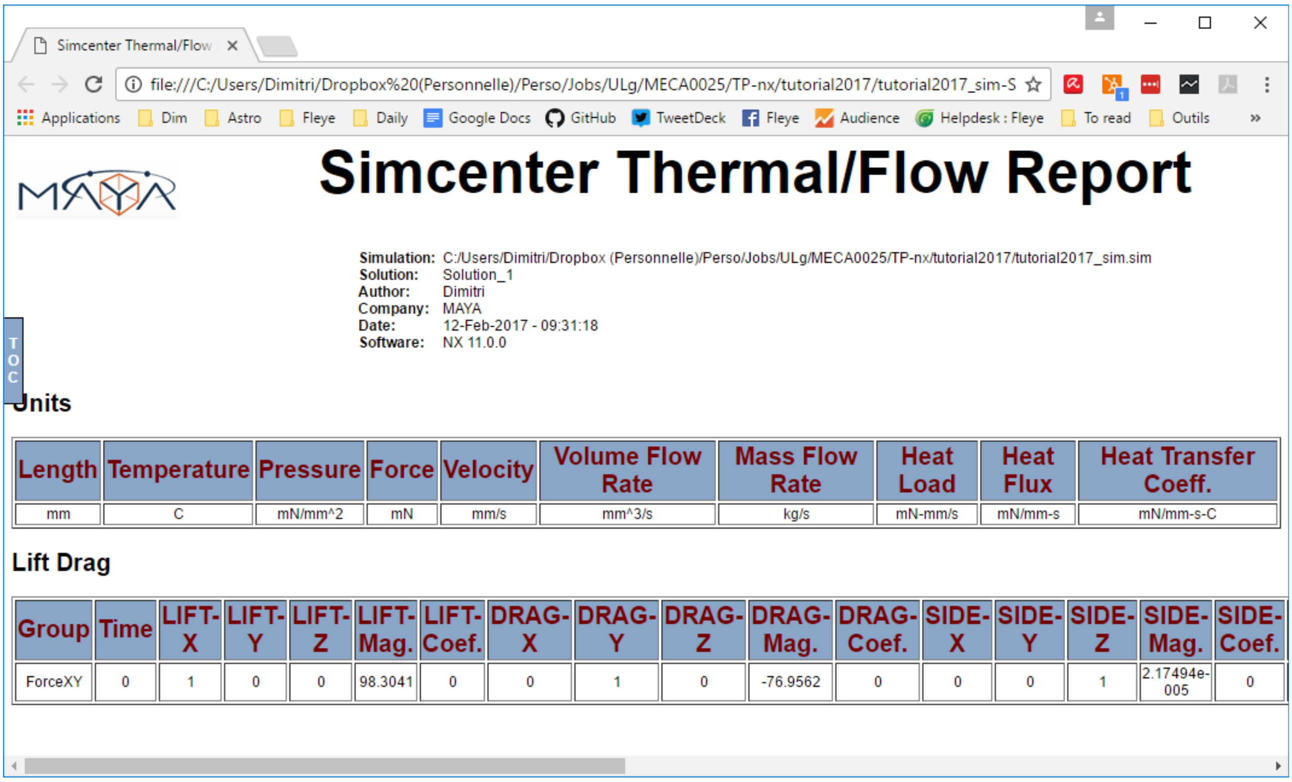


Click Yes to review all the solver verbose. There is very valuable information to understand better the solution and how worked the solver.

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.



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

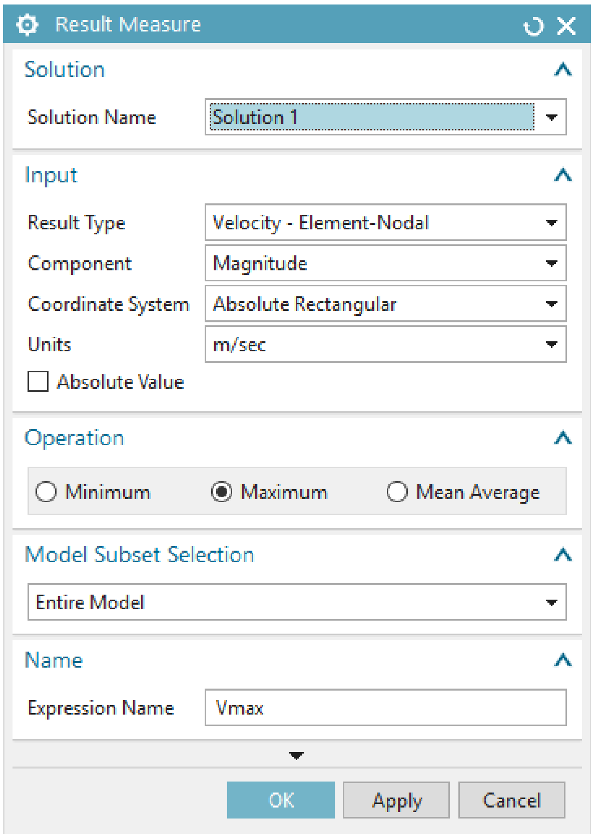
5.3 Set-up key measurements for rapid analysis

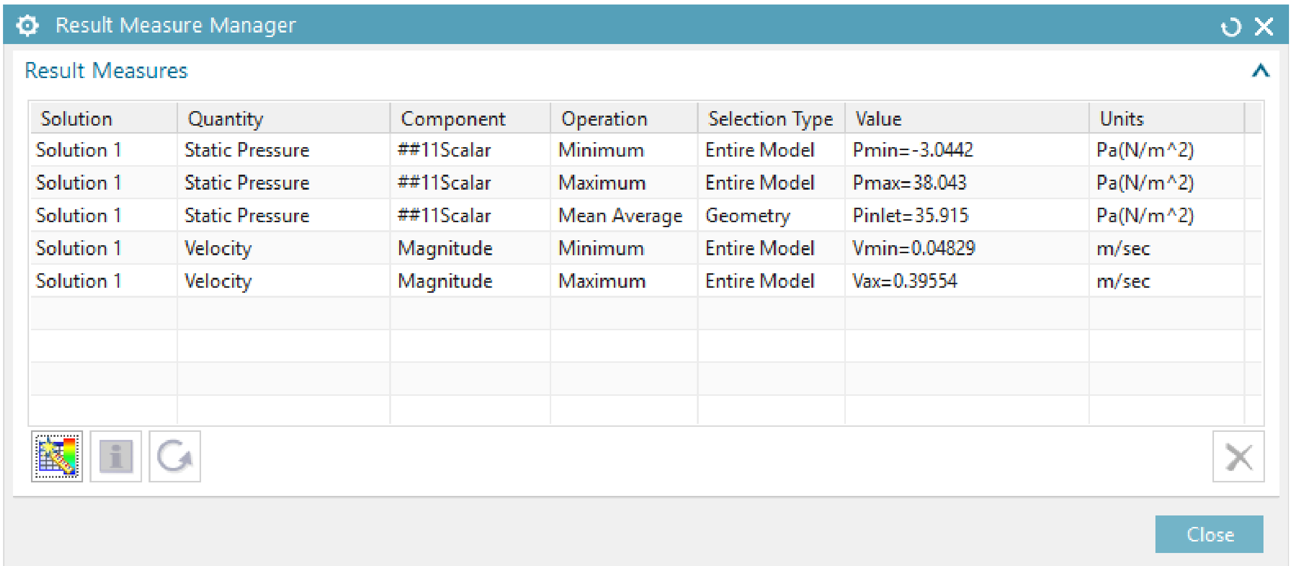
Once you have solved at least one time the simulation, you can add some key measures of interest. These values are handy for a quick look on the solution when varying different solver parameters or trying different mesh configuration.

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.





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

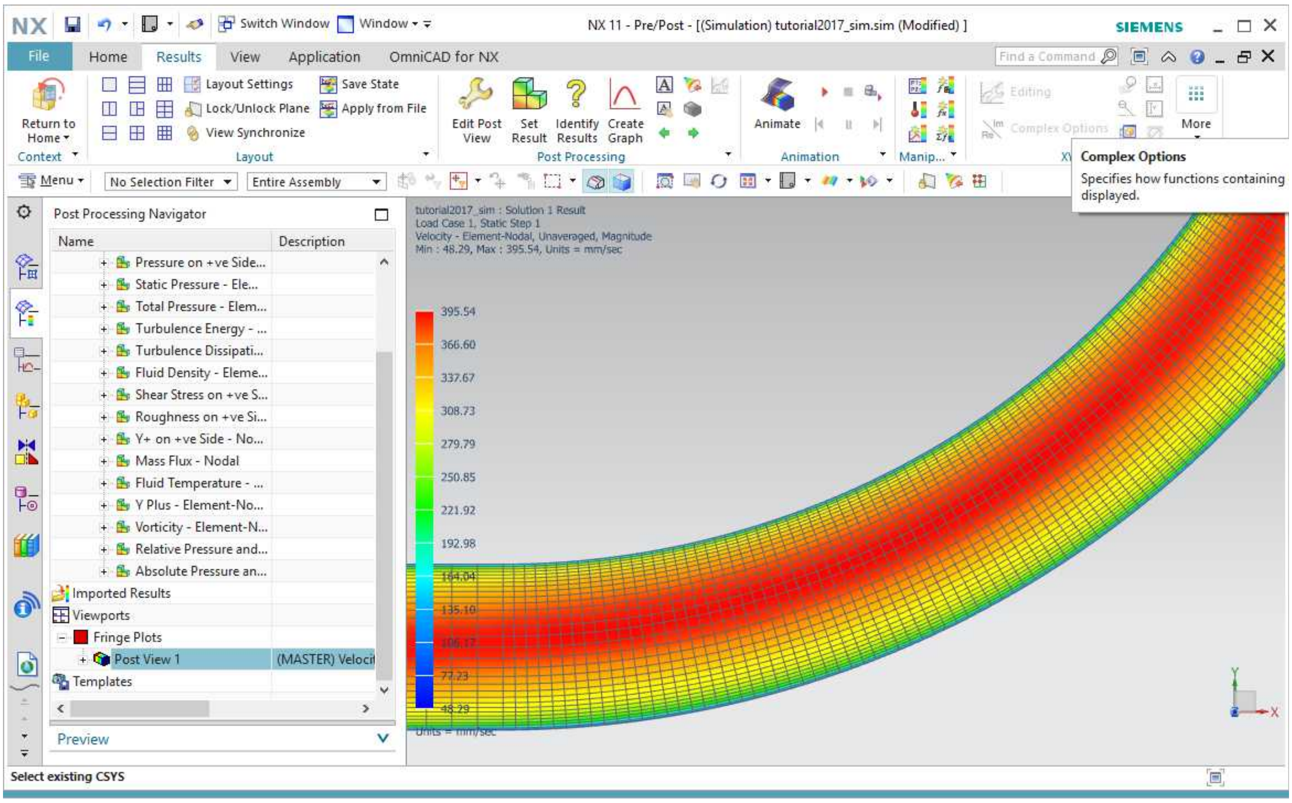
5.4 Plot the results

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 iso-curves, 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

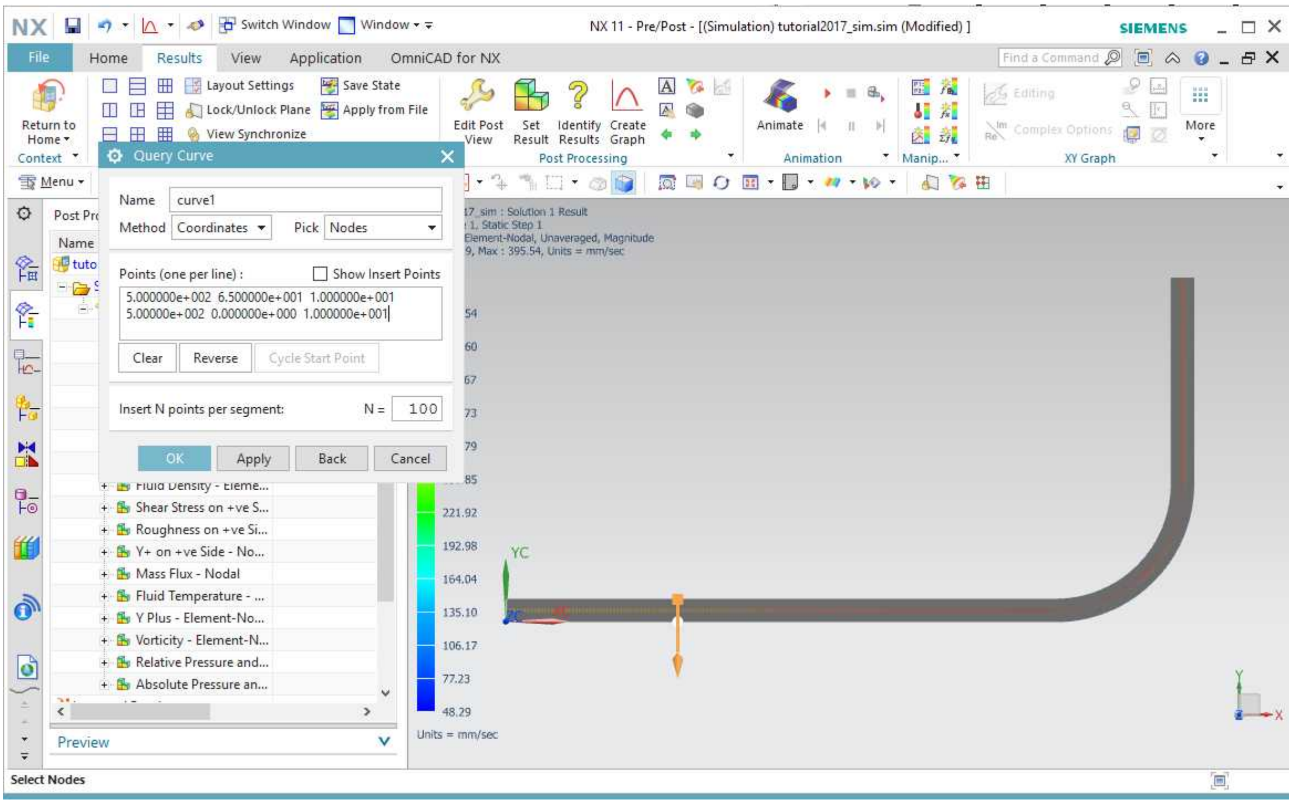
Under the “Query curve” tab, click the top right icon to create a new curve.

Give a name to the curve and select Method “coordinates”

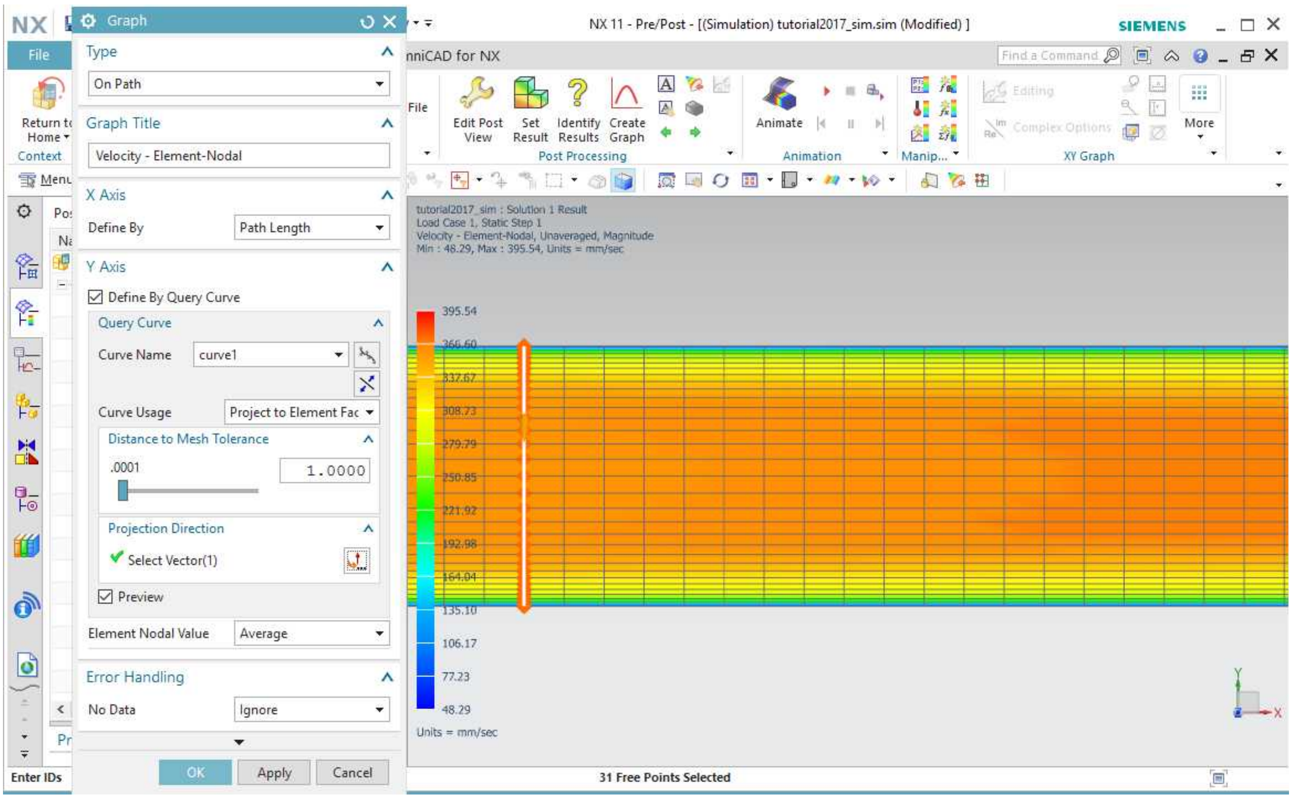
You can then either click the two extremities of the curve on the mesh display or enter the points coordinates manually.

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.

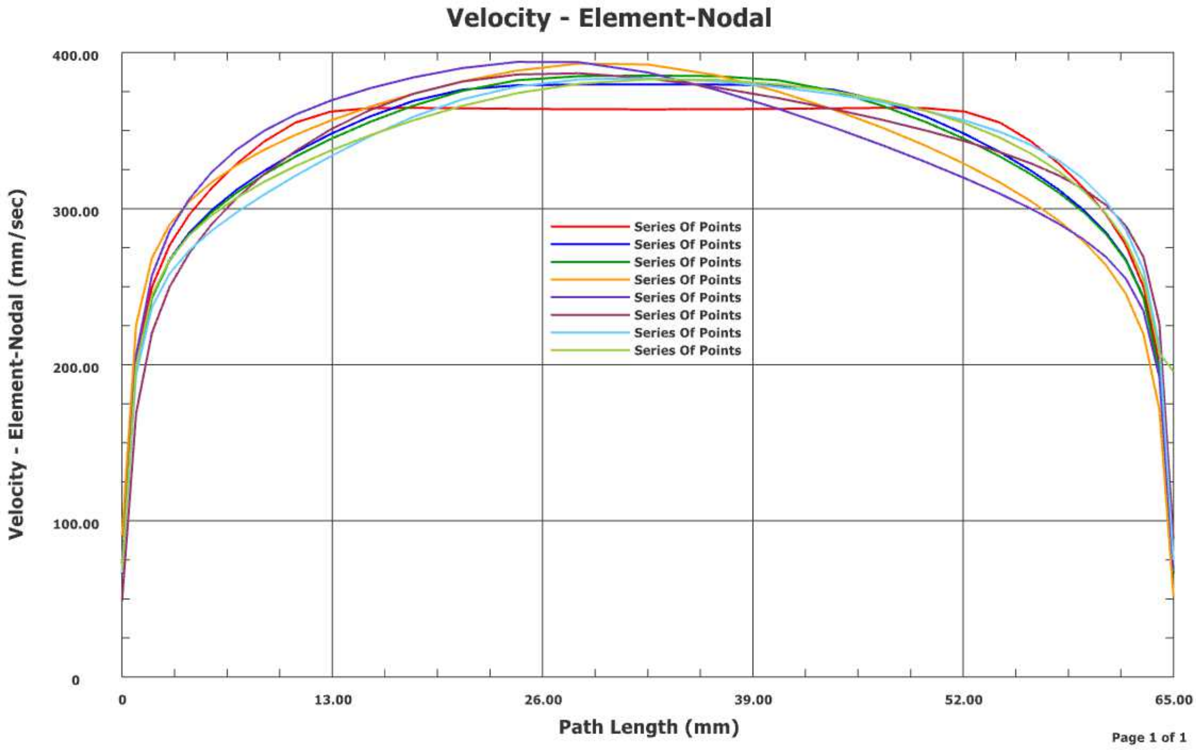


When back in the graph tab, make sure the slider for “Distance to Mesh” is at the minimum 0,0001 and select the Projection Vector to be along z-axis.



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.



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/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” 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.

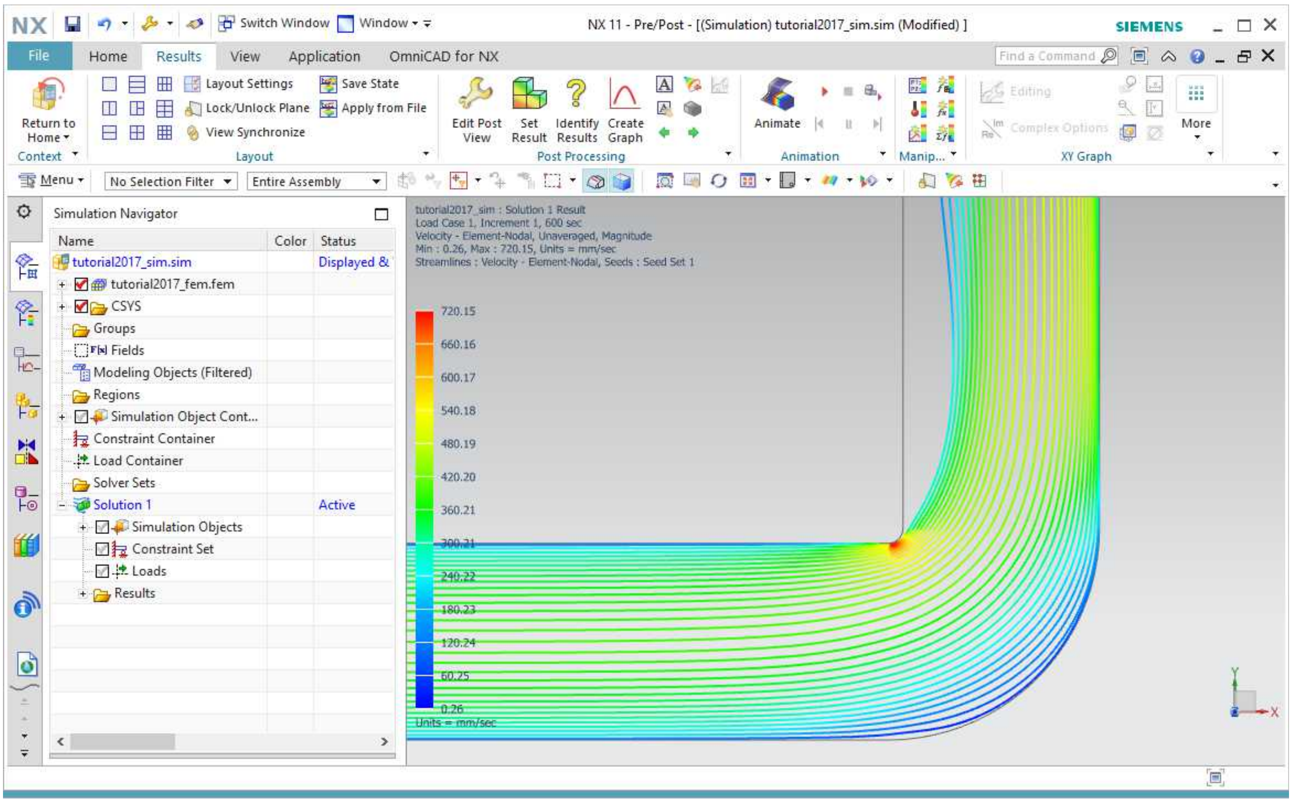
In the top-left, click “Activate Meshing”.

In the top-left, click Update, to automatically update the meshing to the new geometry.

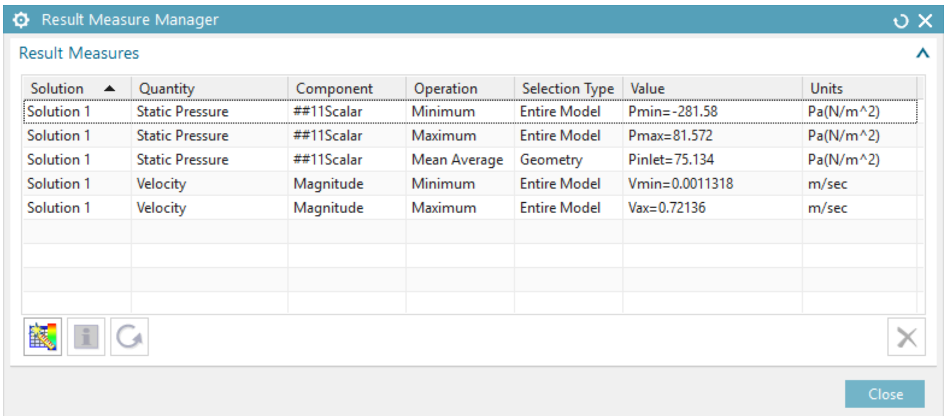
Then again in the top left “Activate the Simulation” and Solve for this new configuration.

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.



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.


Result Measure Manager

Result Measures

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

Close

	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//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/11/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