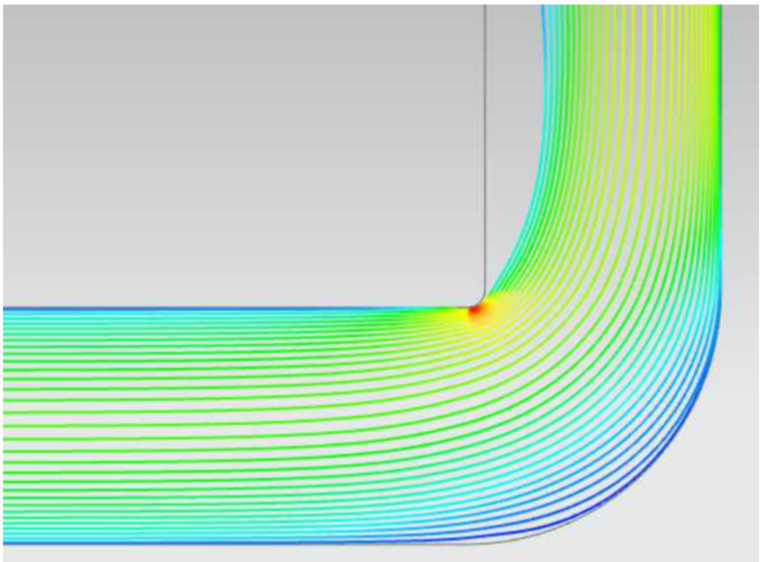
February 2021

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

Summary and work-flow

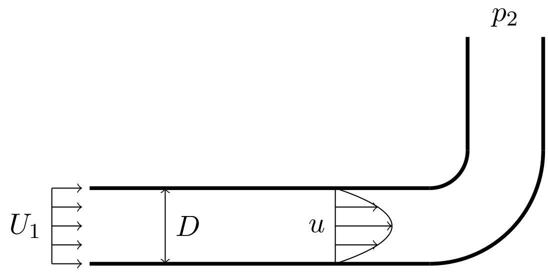
This tutorial explains the work-flow to set-up and run a 2D Flow Simulation in Simcenter NX12. Its targeted audience is anyone without any particular prior knowledge of NX Simcenter Environment. It explains in details the basic actions in order to perform a simple fluid simulation. It is meant to be 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 `yourproject.prt` 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 (`.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),

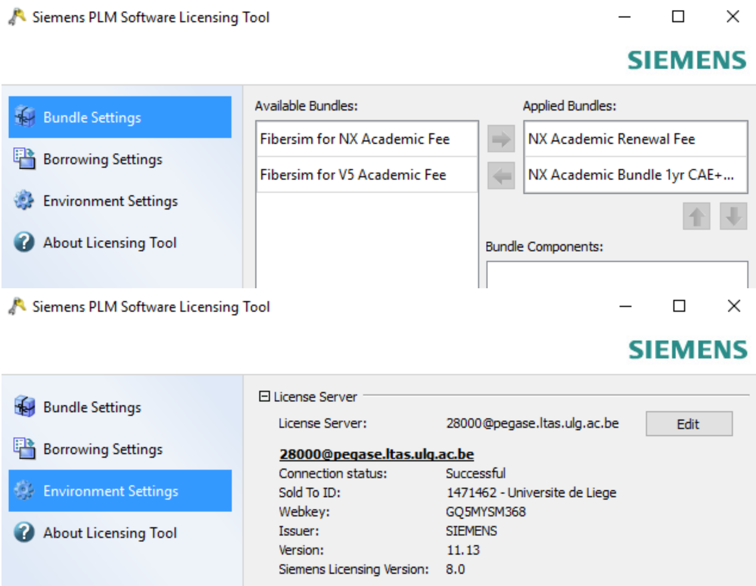
$$\Delta p = f \frac{L}{4D} \rho U_1^2$$

with f the Darcy friction factor, obtained (experimentally) for a turbulent flow in a smooth pipe (here, $Re = \frac{U_1 D}{\nu} = 2.2\ 10^4 > 2000$ 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

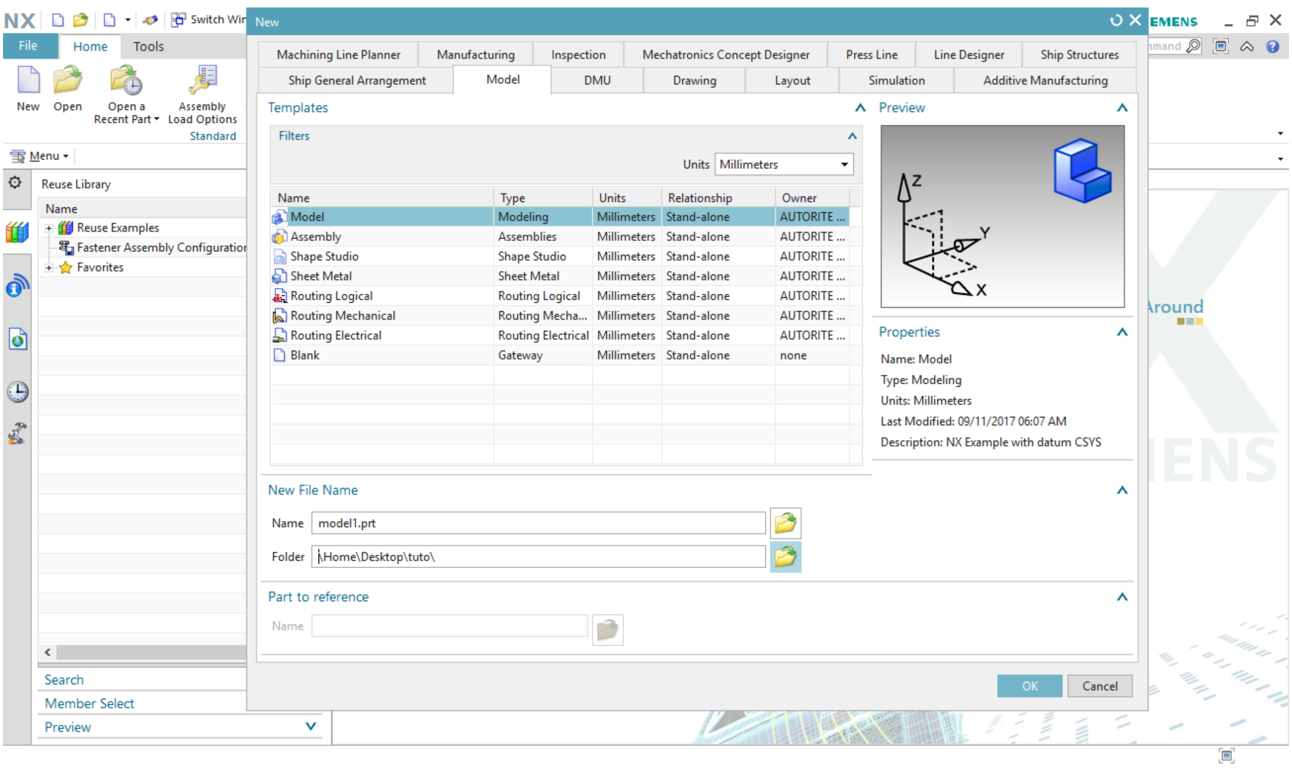
Before to open NX12, be sure that you are connected to the network of ULiege (via the VPN BIG-IP Edge CLient). If you connect for the first time, type *Licensing Tool* in Start Menu and select these bundles. Afterwards, you can start NX12.



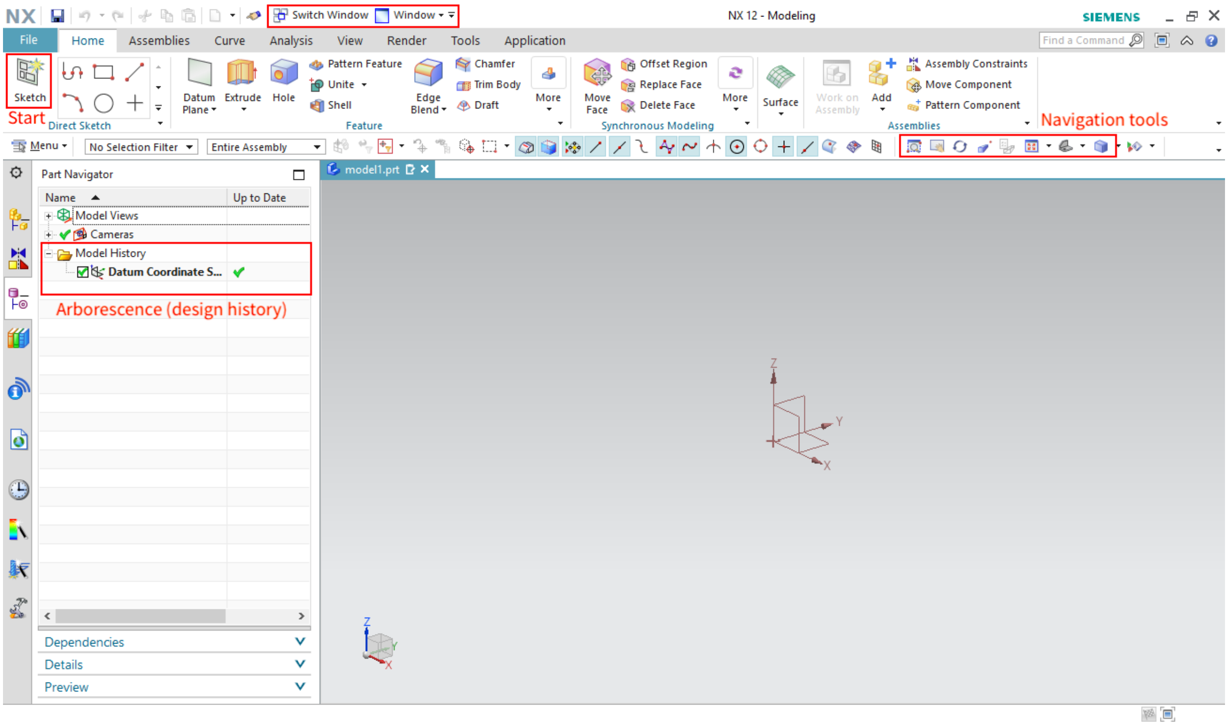
1.1 Create an empty model

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.



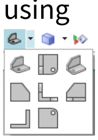
You shall have this view and you are ready to create your part.



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* Switch Window to navigate.

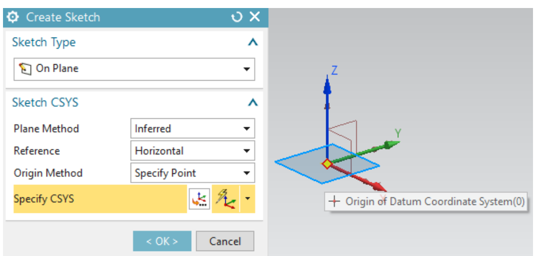
To **navigate in space**, you can use rather navigation icons or using a mouse :

— 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

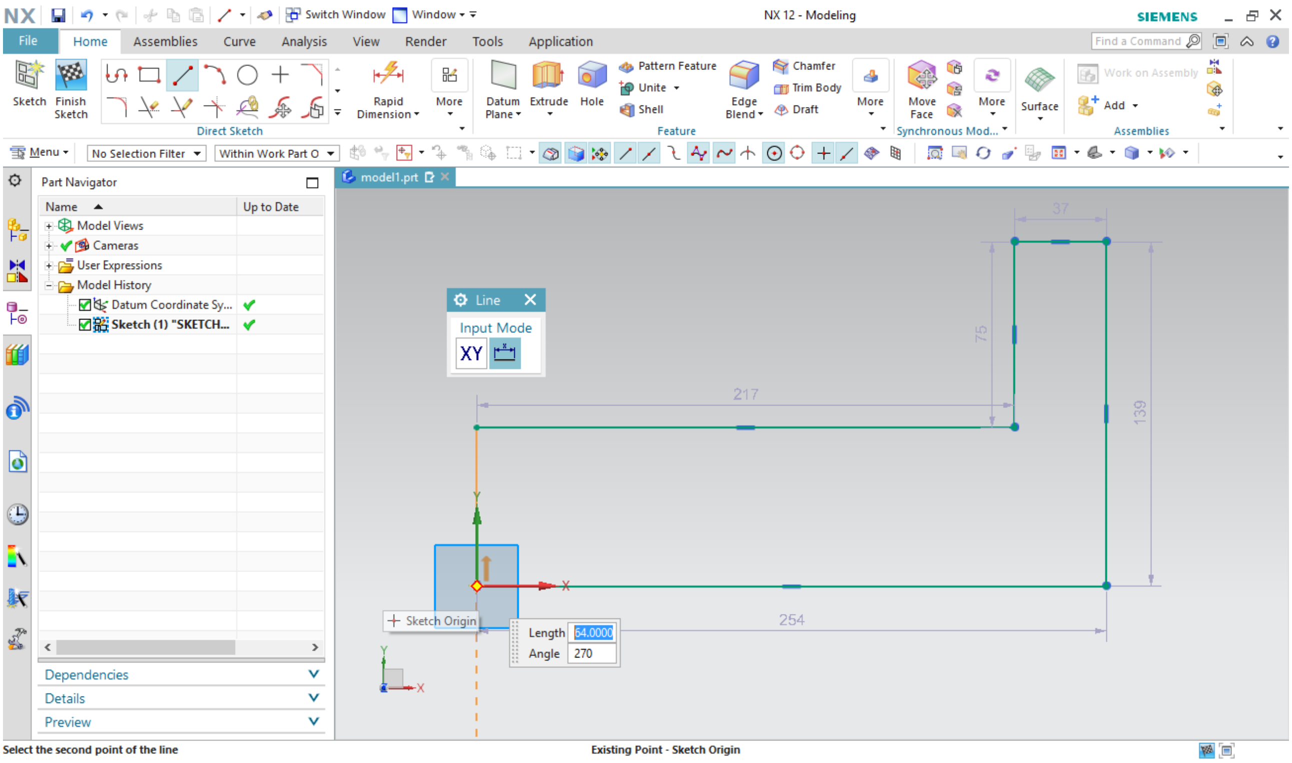
Click “Sketch” in the upper left corner Sketch.
You can keep the default coordinate axis > *OK*



1.3 Draw the fluid domain

Draw the fluid domain and eventual obstacles using the tools under Direct Skecth : Rectangle, Line, Arc,... 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* command. The dimensions do not have to be the same at this moment, only the general shape matters.



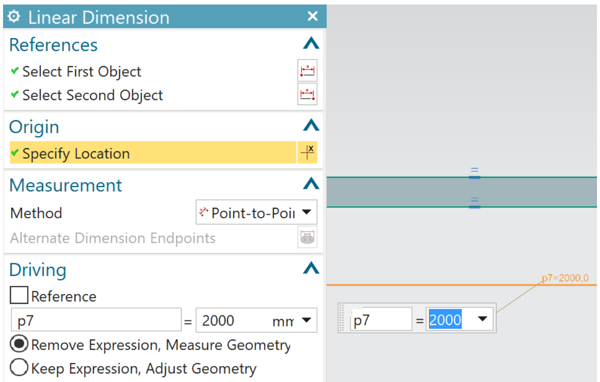
Closed sketch : To be sure that your sketch is a closed contour (to be able to create a solid body in section 1.5), be sure that you select the same point when drawing intersecting lines (orange point). NX12 helps you to draw vertical or horizontal line by adding some orange dashed lines.

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 ✕ (or click on the element and CTRL+D).

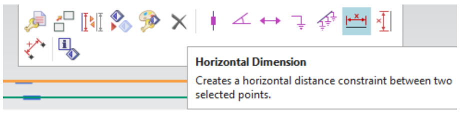
Element selection : The different elements are selected just by clicking them successively, there is not need to hold down CTRL key. To deselect all : press ESC. To deselect one item only : click it with left mouse holding down SHIFT

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

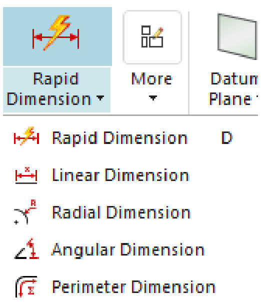
1. Double-click on grey dimension and adjust the value to make them constraints, they will appear in dark blue color.

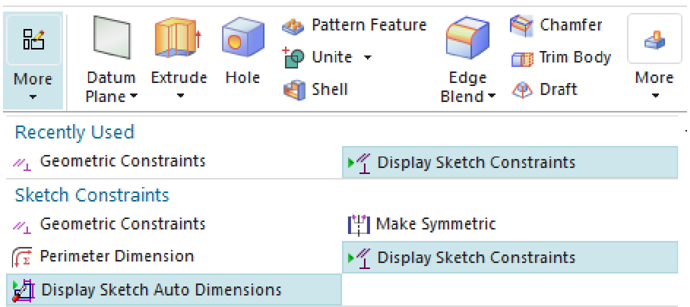


2. Select a segment, right click and then choose *horizontal/vertical dimensions*

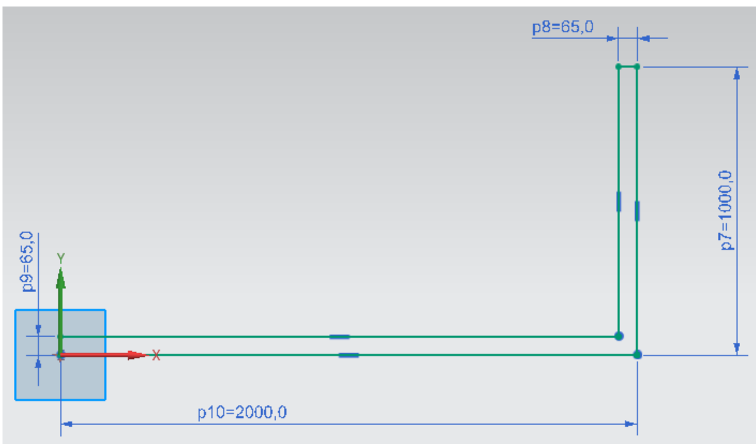


3. Add more complex constraints, like parallelism, same length, perpendicularity, etc.. by selecting successively different elements and then choosing suggested constraints.
   Dimensions constraints can be found under *Direct Sketch* > *"Rapid Dimension"*. You can also find all constraints under *Direct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





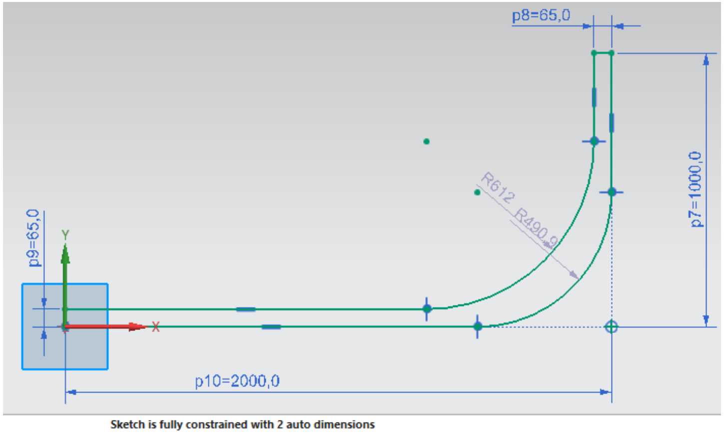
For this tutorial you should have the following constraints.



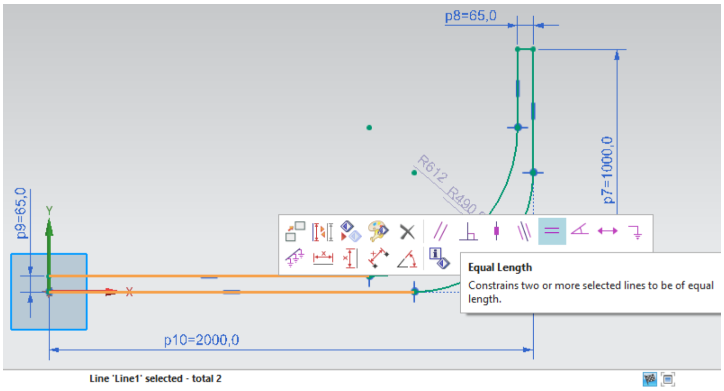
Sketch is fully constrained

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.

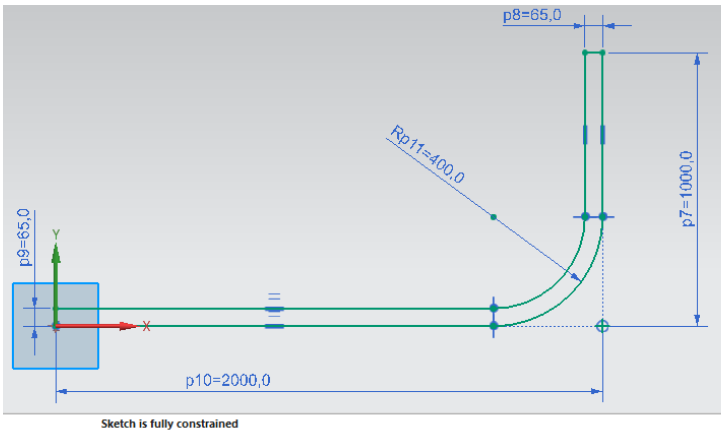
Draw a round corner : Select the *Filet* tool  in *Direct Sketch* and then successively the two inner legs of the corner. Repeat with the two outer legs. You should have the following sketch with two additional degree of freedom, being the two arc radius.



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.

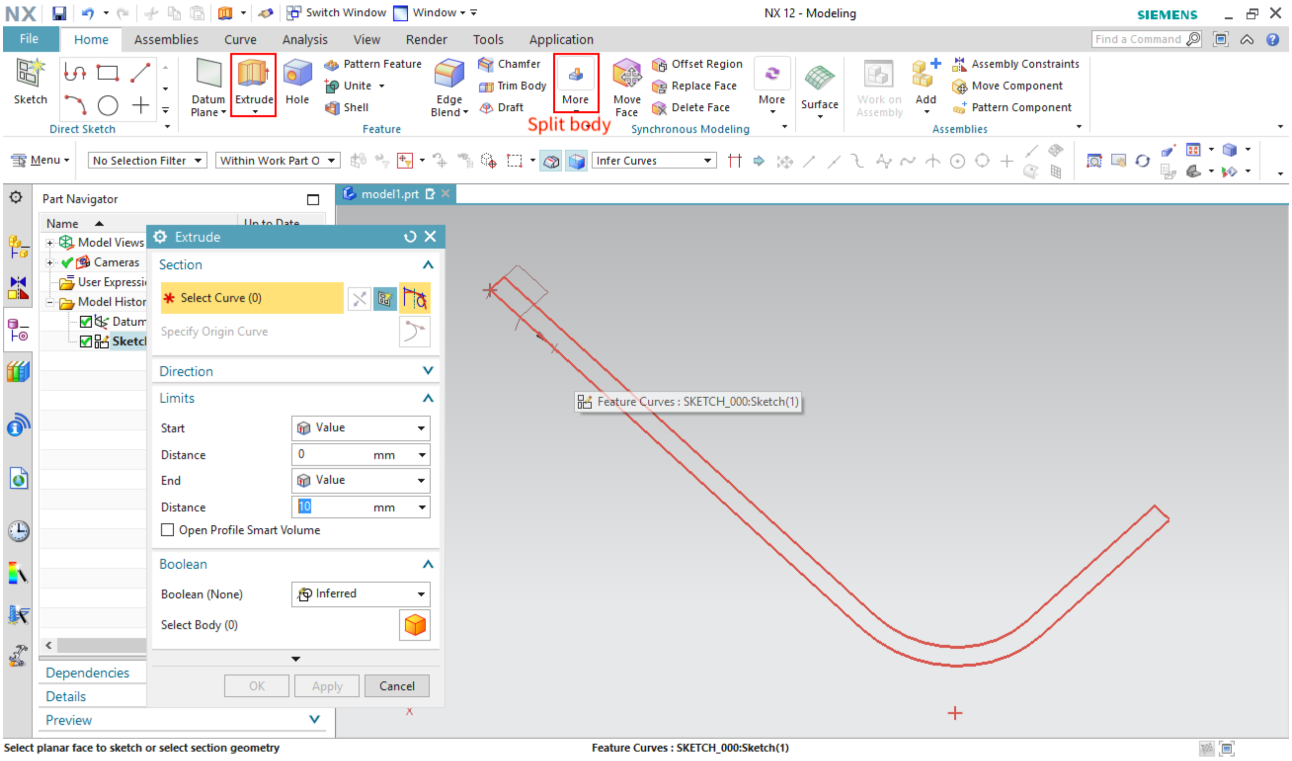


1.4 Finish the sketch

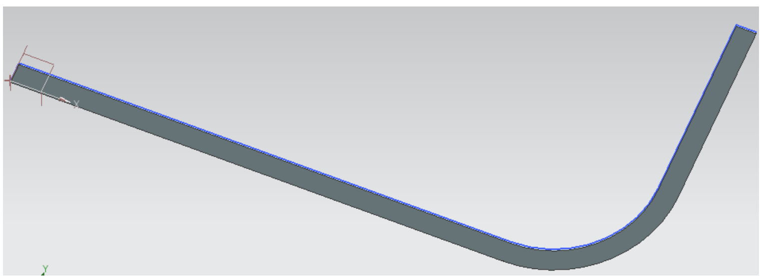
When you are done with the sketch, *Click "Finish Sketch" in the upper left corner*

1.5 Extrude

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



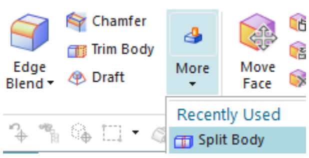
By rotating the view with the middle mouse button you can verify that you have now a 3D solid body (if you did not have close your sketch, it will be a shell).



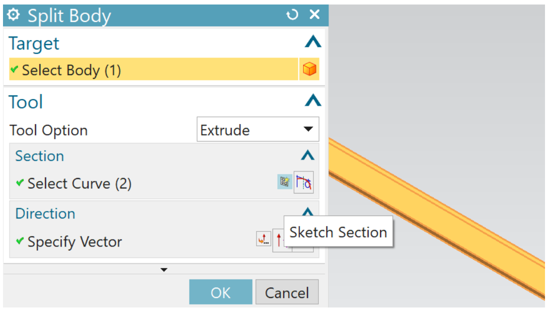
1.6 Split body to prepare the meshing

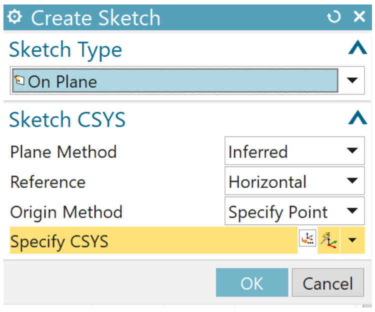
To make a consistent meshing with quadrilateral elements, and be able to apply specific mesh constraints on domains, the fluid domain has to be divided in simple geometries : triangles or quadrilaterals, possibly with curved edges. Thus, we will split here the body in 3 parts thanks to two small edges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

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

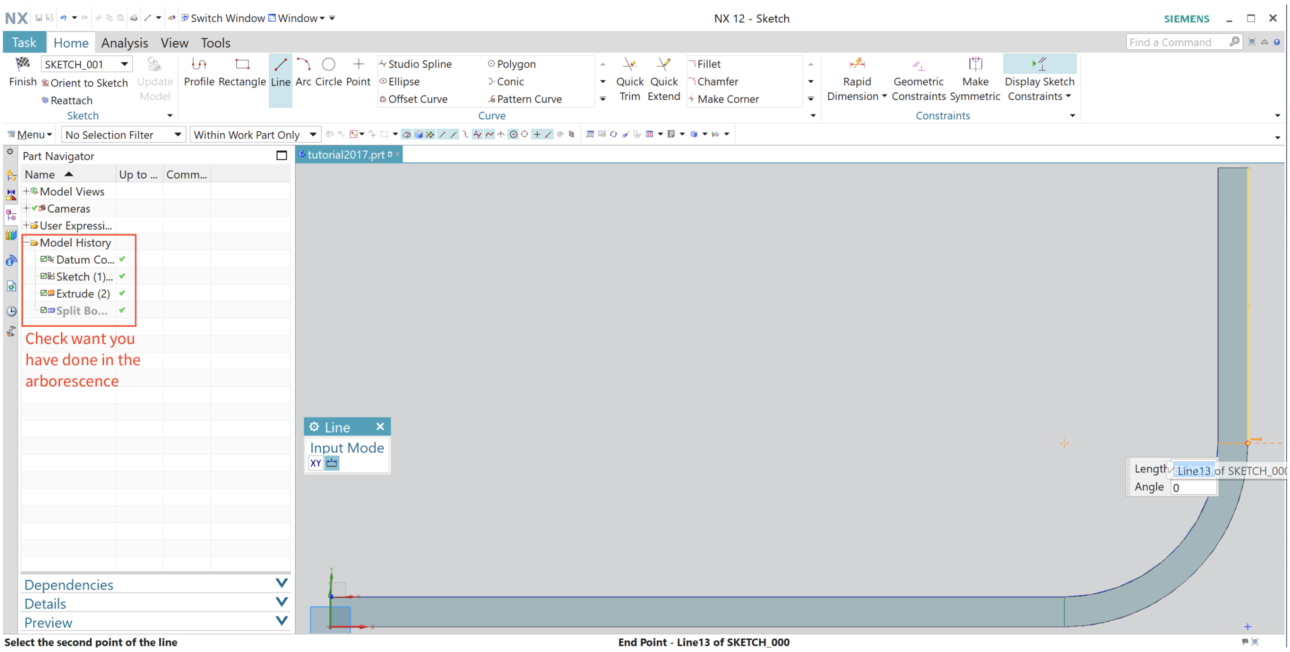


Select your body, choose *"Extrude"* under *Tool Option* and then under *Tool > Section > Select Curve > Click on "Sketch Section"*.

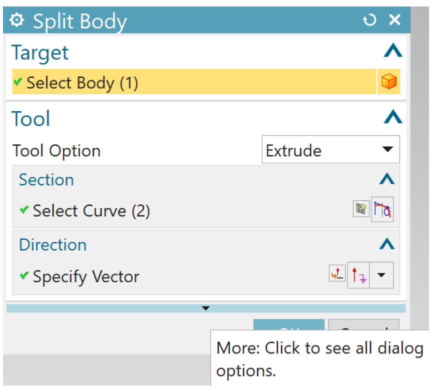


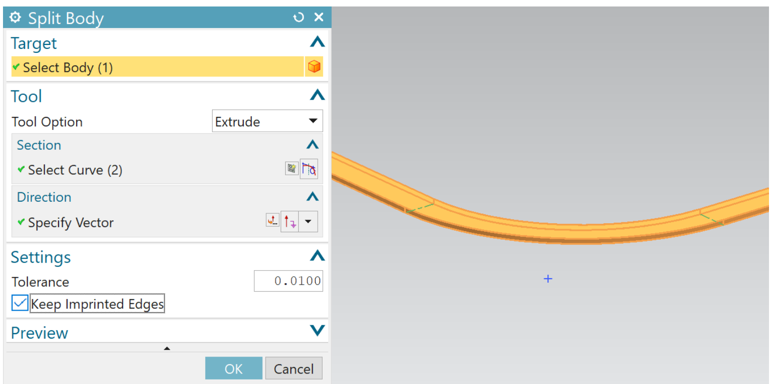


Draw split limit : After clicking on *"Sketch Section"*, you will enter in a temporary sketch, where you have to first select the system coordinate (same as before, CSYS).
You should split the body in three parts by adding two perpendicular lines at the entry and exit of the corner. You should also have this new sketch fully constrained. When you are done with the sketch, Click *"Finish Sketch"* in the upper left corner.

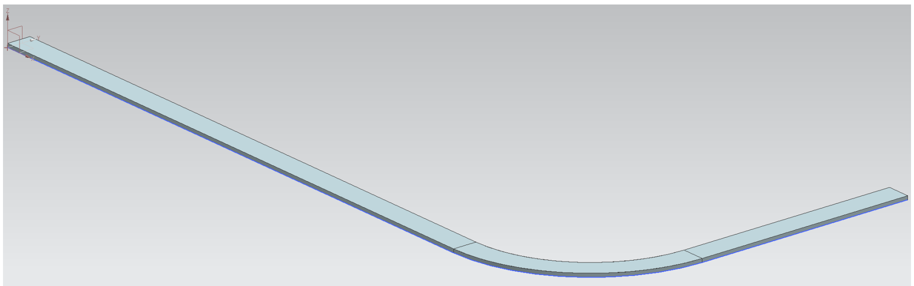


Make sure to tick the box for *"Keep Imprinted Edges"* under *Settings* and click OK. By doing so, the software automatically creates Glue Coincident type mesh mating conditions between the bodies when you switch to the FEM file. You should inspect these mating conditions and ensure that they were created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





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

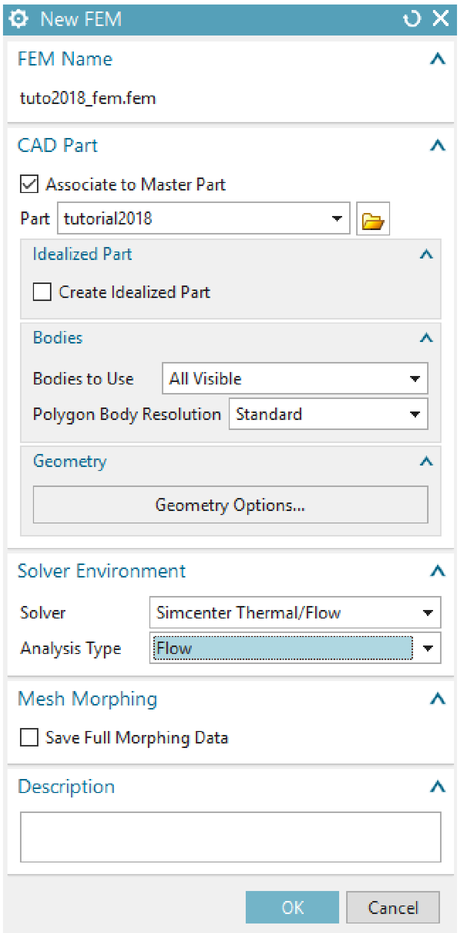
— *File > New > under tab "Simulation" select "Simcenter Thermal/Fow" with type "Fem" > OK* Be sure to save it in the same folder as the `.prt` file, under another name (using `name_fem.fem` for example).

— Keep the box *"Associate to Master Part"* ticked.

— In the field *"Part"*, you shall select the part you just created.

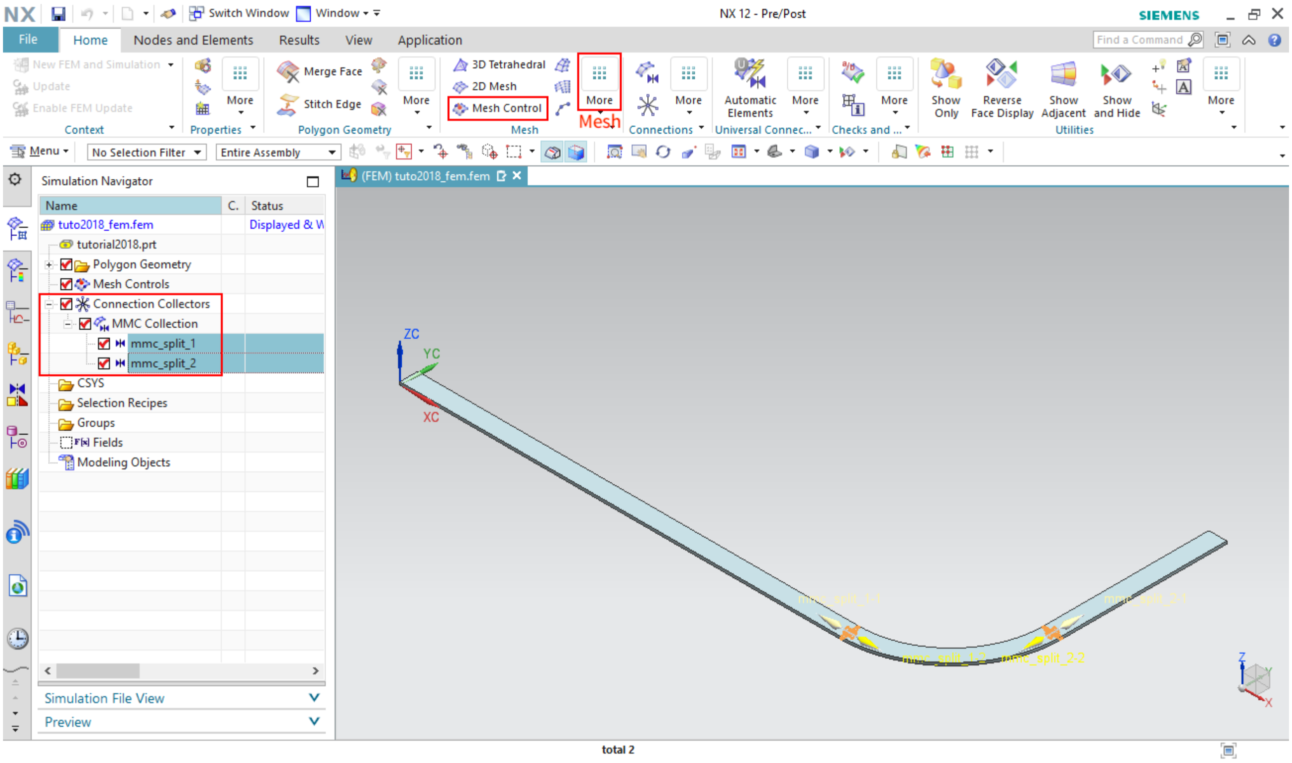
— Untick the box *"Create Idealized Part"*. This option is used when working in team on the same part or if the part has to be simplified for the simulation but the original design has to kept unaltered.

— Select *Solver "Simcenter Thermal/Flow"* and *Analysis Type "Flow"*. Click *OK*.

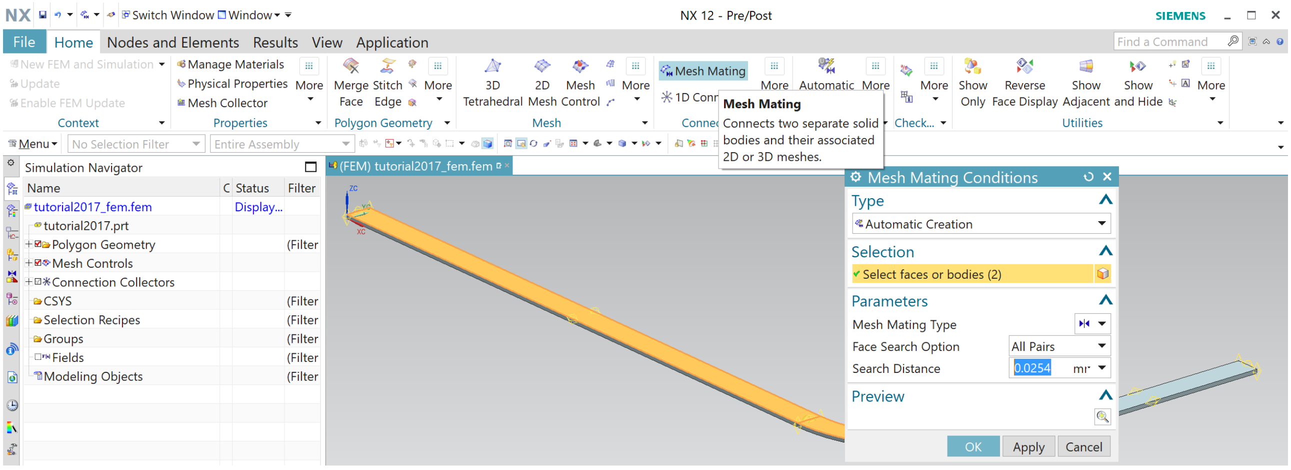


2.2 Verify that the different split bodies are connected

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).

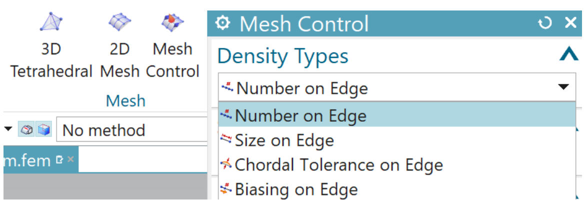


If it appears that two polygon bodies have common face but are not connected, you can add the connection manually by clicking Mesh Mating under Connections toolbox.



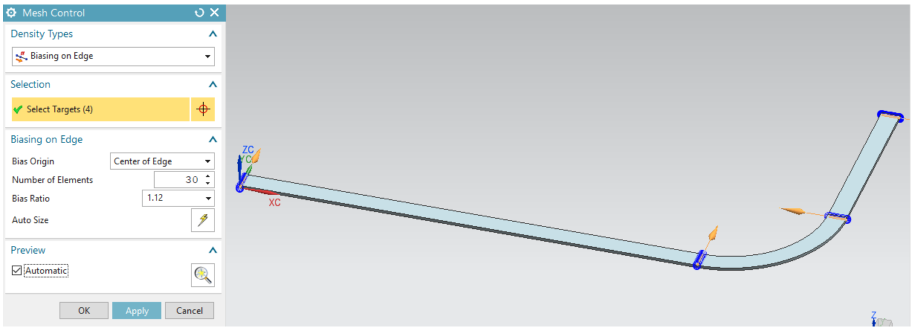
2.3 Add the meshing constraints on the body edges

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.

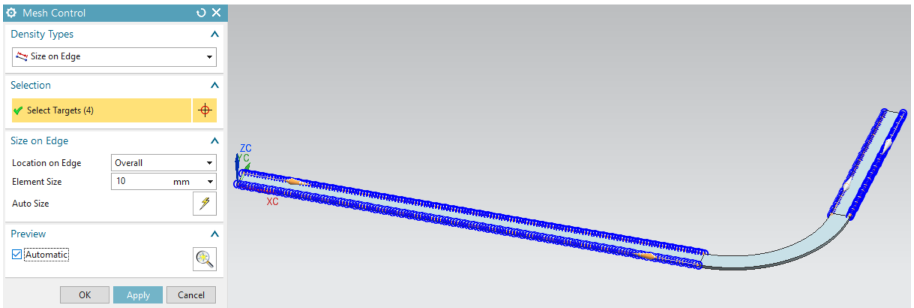


Here, we will set the mesh constraints, on one face only (upper face of the extrude, be sure to select always edges from the same face).

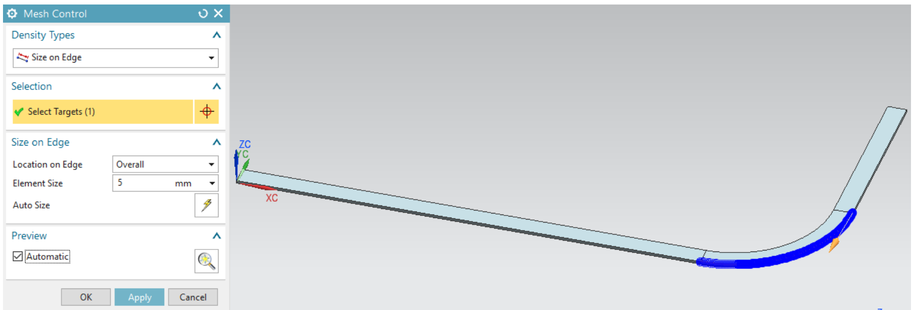
Biasing on edge : we will apply on the 4 perpendicular section a “bias on edge” with option “Center of edge”, 30 elements and bias ratio of 1.12 to have smaller elements close to the wall where the velocity gradients are more important. The orientation of the orange arrow does not matter here because 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.



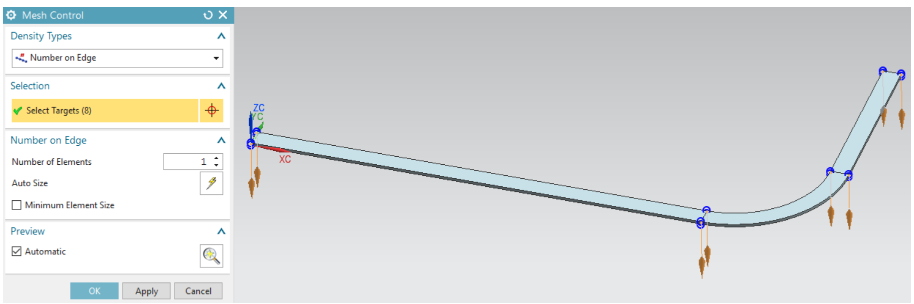
Size on edge : Next, you force the elements size to be 10mm on the entry and exit lateral sections. You shall put all those constraints in the same plane, here the upper plane.



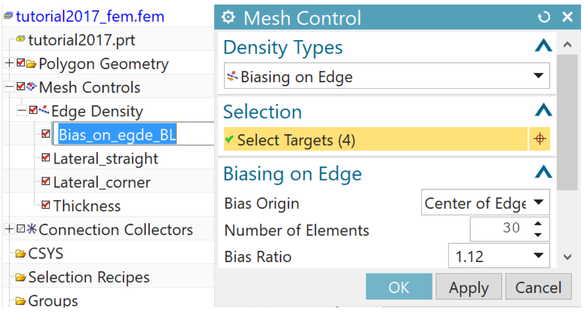
Repeat the same operation and set a size on edge of 5mm on the outer arc of the corner.



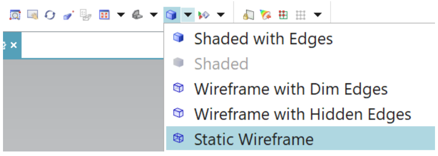
Number on edge : you shall also set the short edges along the thickness (z-axis) to have one single element, as the simulation is 2D. Do not hesitate to zoom a lot on the thickness and to rotate the view to be sure only the small thickness edge is selected.

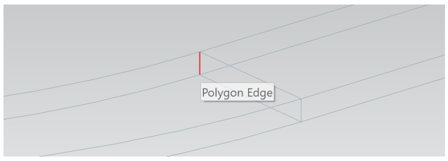


Tips : You can set a constraint on multiple edges at a time. For the biasing on edge (especially useful for boundary layer), you have to select the orientation (start of edge, end of edge or middle of edge) and it varies from edges to edges depending on the edge natural direction.
You can always modify constraints already defined by double click on it in the arborescence. You can also rename the constraint to identify it more easily.



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.

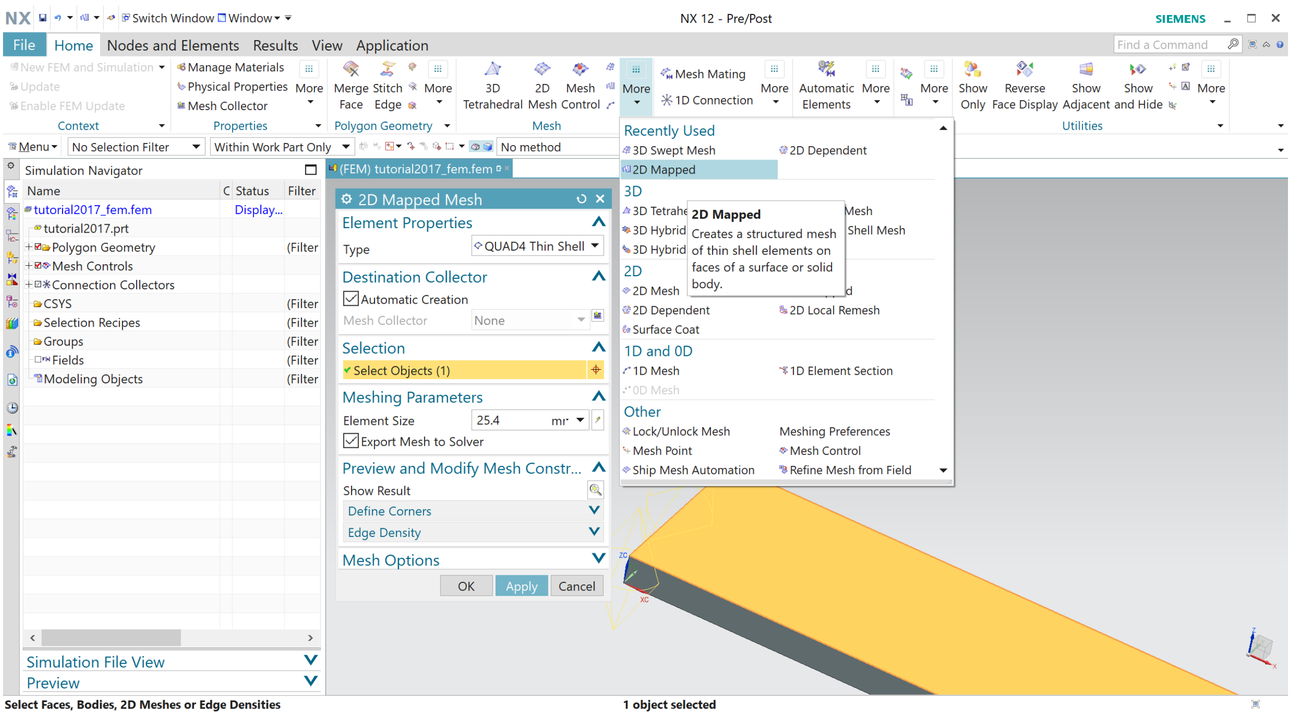




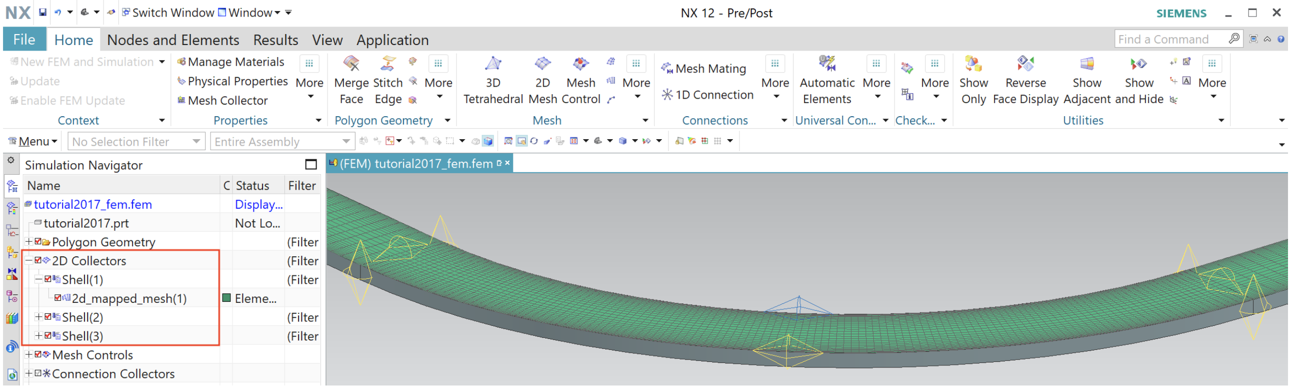
2.4 Create 2D meshes

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.



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.



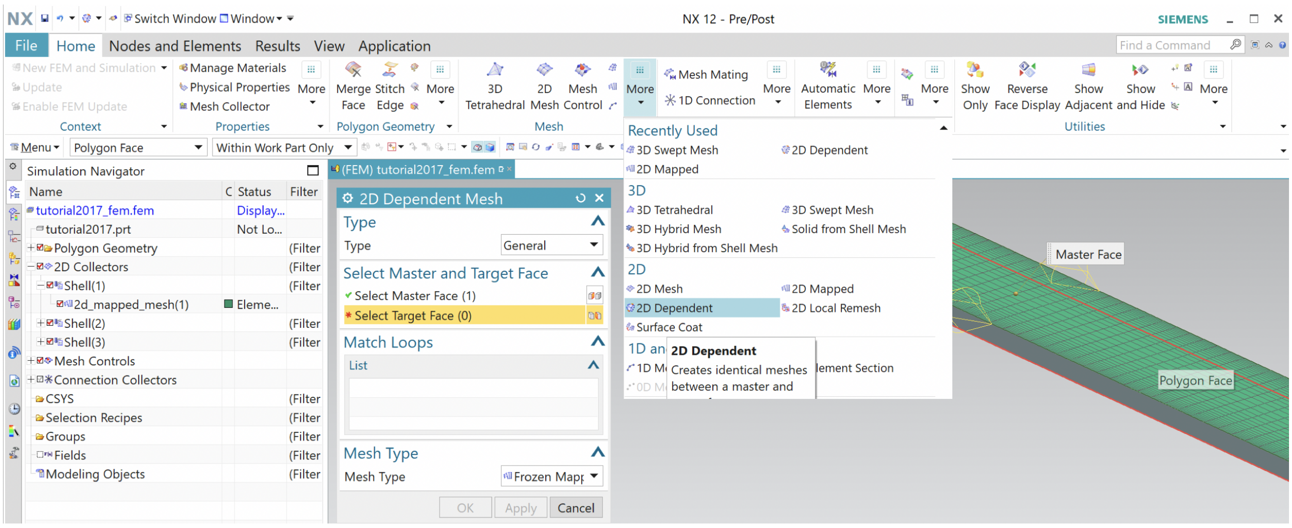
Every time you generate a 2D mapped mesh with quadrilateral elements, the constraint of one edge is reported to the opposed edge. This constraint appears as “Mapped Mesh Edge Density” in the arborescence and as blue diamond in the main view (see image here- above). This also means there might be a logical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



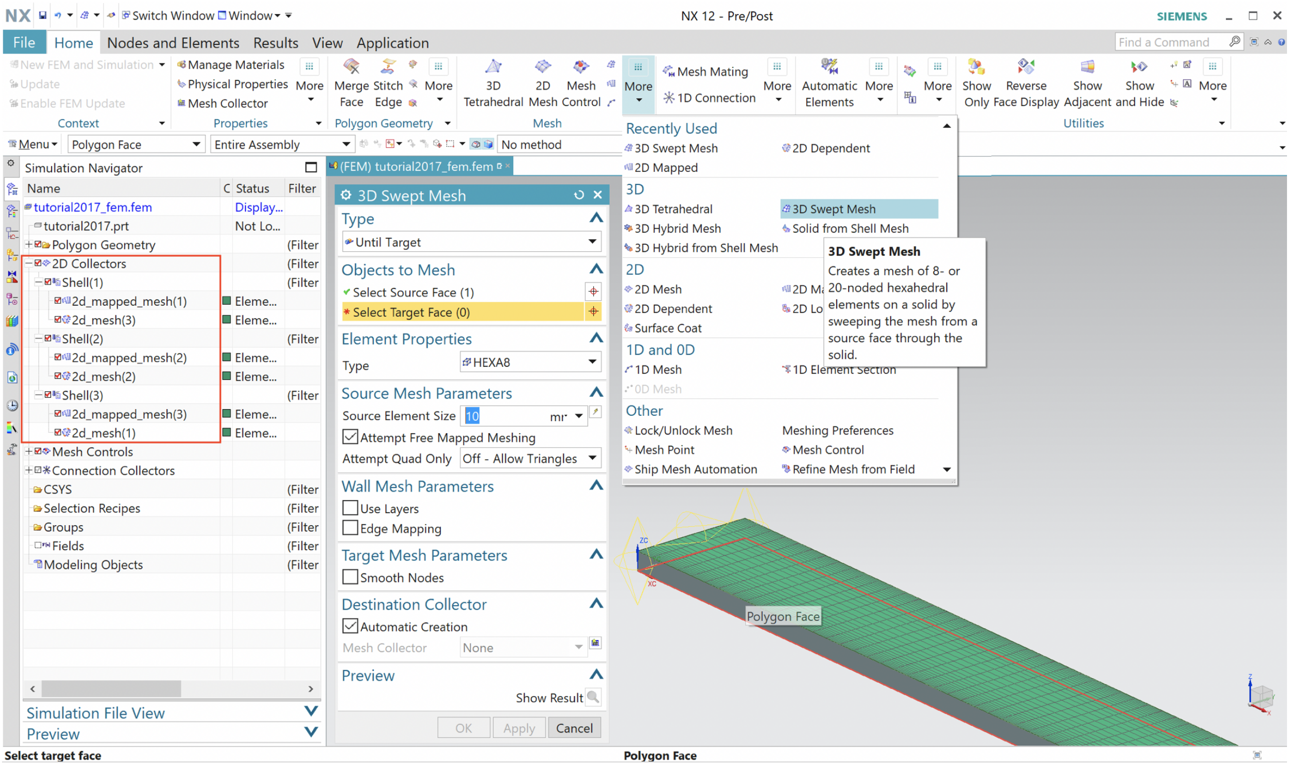
Now, all 3 shells (below *"2D Collectors"*) contain two 2D meshes (the *"2D Mapped"* and the *"2D dependent"* ones, see the figure below in section 2.6).

2.6 Create 3D swept mesh

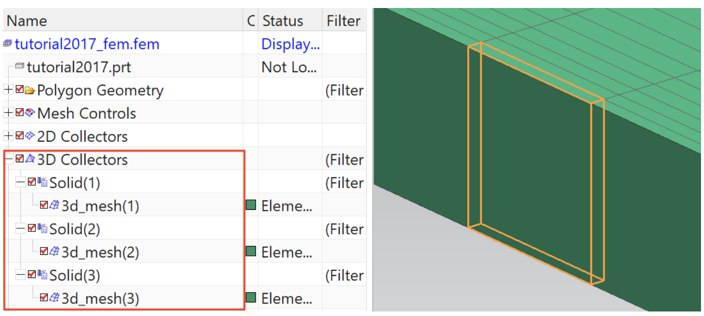
At this stage, when only have 3 shells, each one composed of 2 parallel 2D meshes (upper and lower faces). We will construct the 3D mesh by sweeping the 2D meshes (for the 3 bodies separately). We will now use the mesh constraint along the thickness.

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.



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

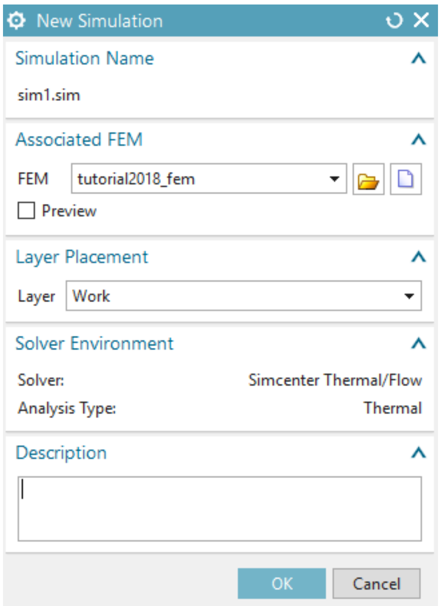
3.1 Create a sim file

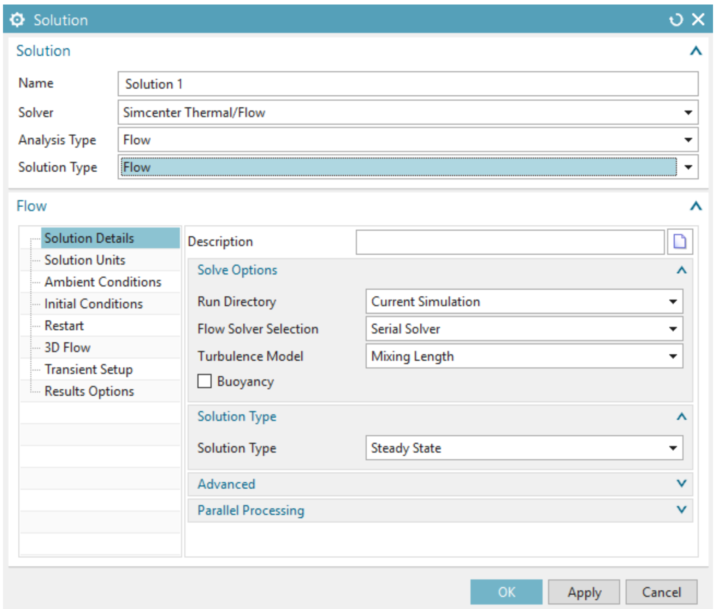
— *File > New > under tab "Simulation" select "Simcenter Thermal/Fow" with type "Sim" > OK*

— 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.

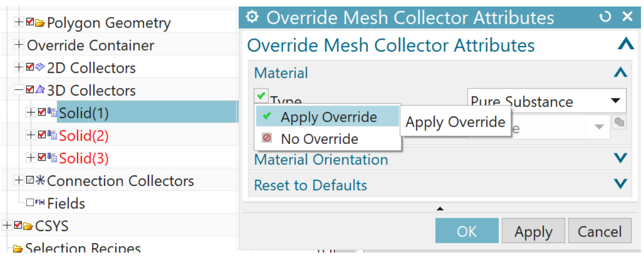
— After changing the *Analysis Type* to *"Flow"*, Click *OK*, as it will still be possible to change all these solution attributes before running the simulation.

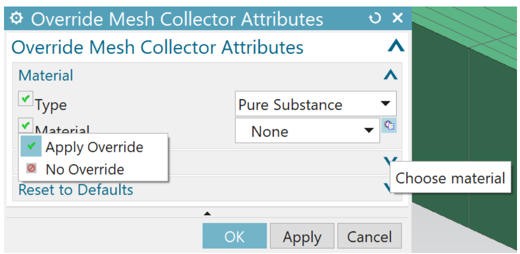




3.2 Set the materials

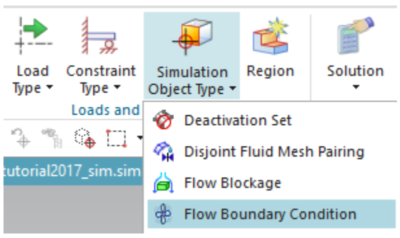
In the arborescence, expand the *"3D Collectors"* and double-click every single *"Solid"*. Then, on the left of *"Type"*, click on the symbol ø to ✓ *"Apply Override"* for both the *"Type (Pure Substance)"* and the *"Material"*. On the right of *"Material (None)"*, select *"Choose material"* and then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



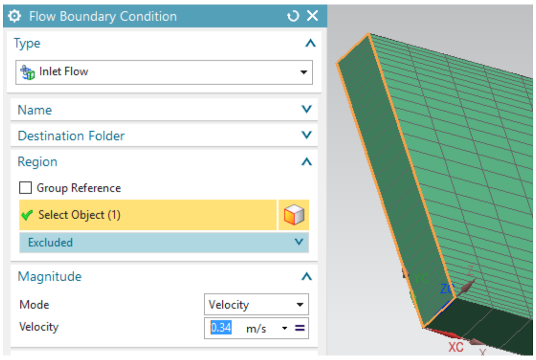


3.3 Set the boundary conditions

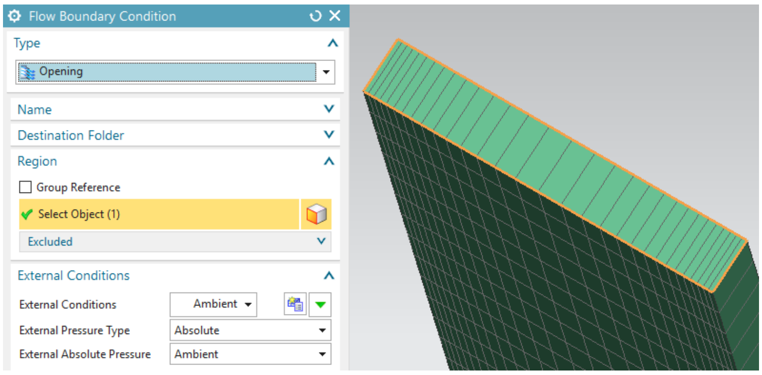
3.3.1 Flow Boundary Condition : Define the inlet and outlet conditions : *Loads and Conditions > Simulation Object Type > "Flow Boundary Conditions"*



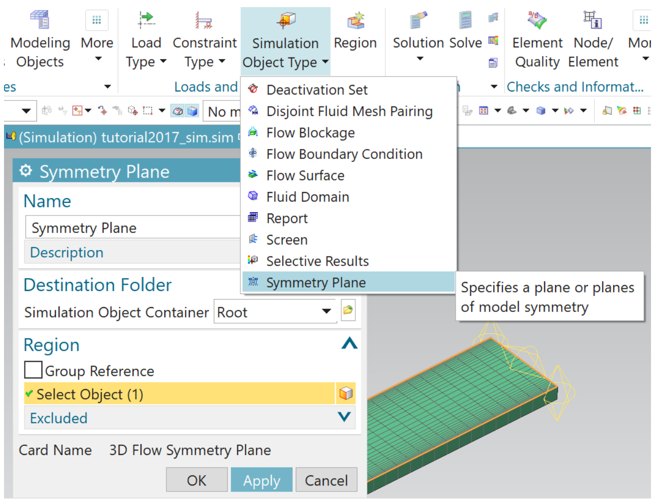
— **Inlet Flow :** to set a velocity (input). In this tutorial, you will set *"Velocity" Mode* to 0.34 m/s (input velocity at the extremity of long leg of the pipe (on the inlet face)).



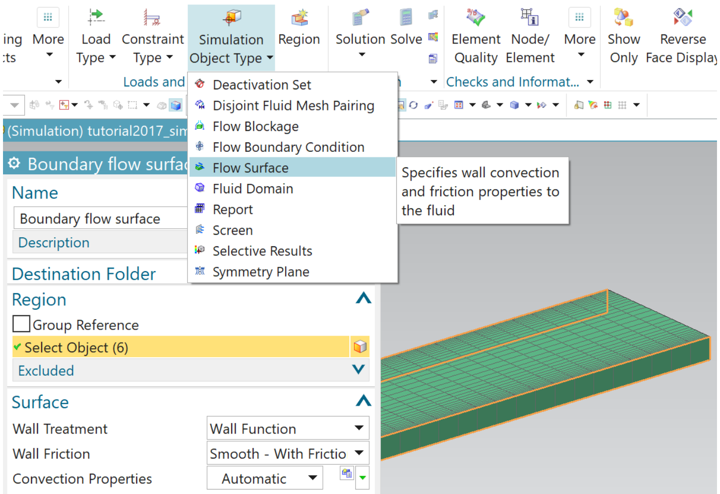
— **Opening :** to set a pressure (output). Select *External Conditions "Ambient"* pressure at the extremity of the short leg (outlet face).



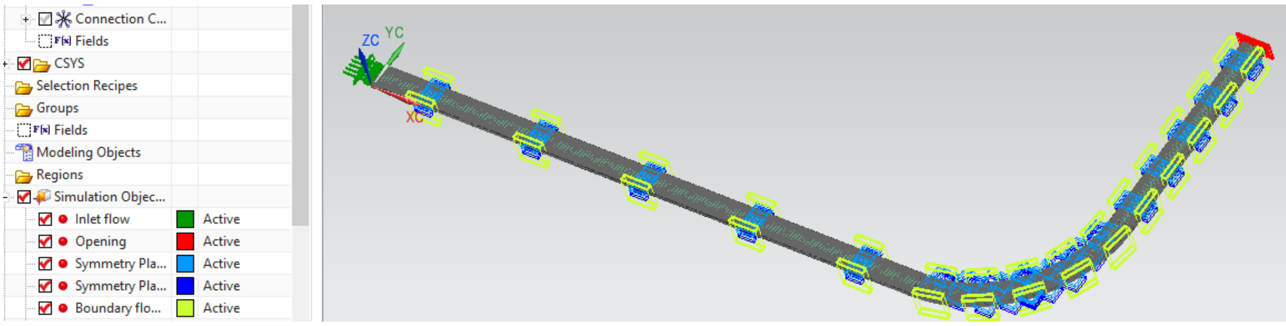
3.3.2 Symmetry Plane : On the upper (and lower) face of the solid, you can impose *"Symmetry Plane"* to ensure a pure bi-dimensional problem. You can select all faces on one side simultaneously (all 3 faces on the upper surface for example then apply and proceed the same for all 3 faces on the bottom 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).



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).

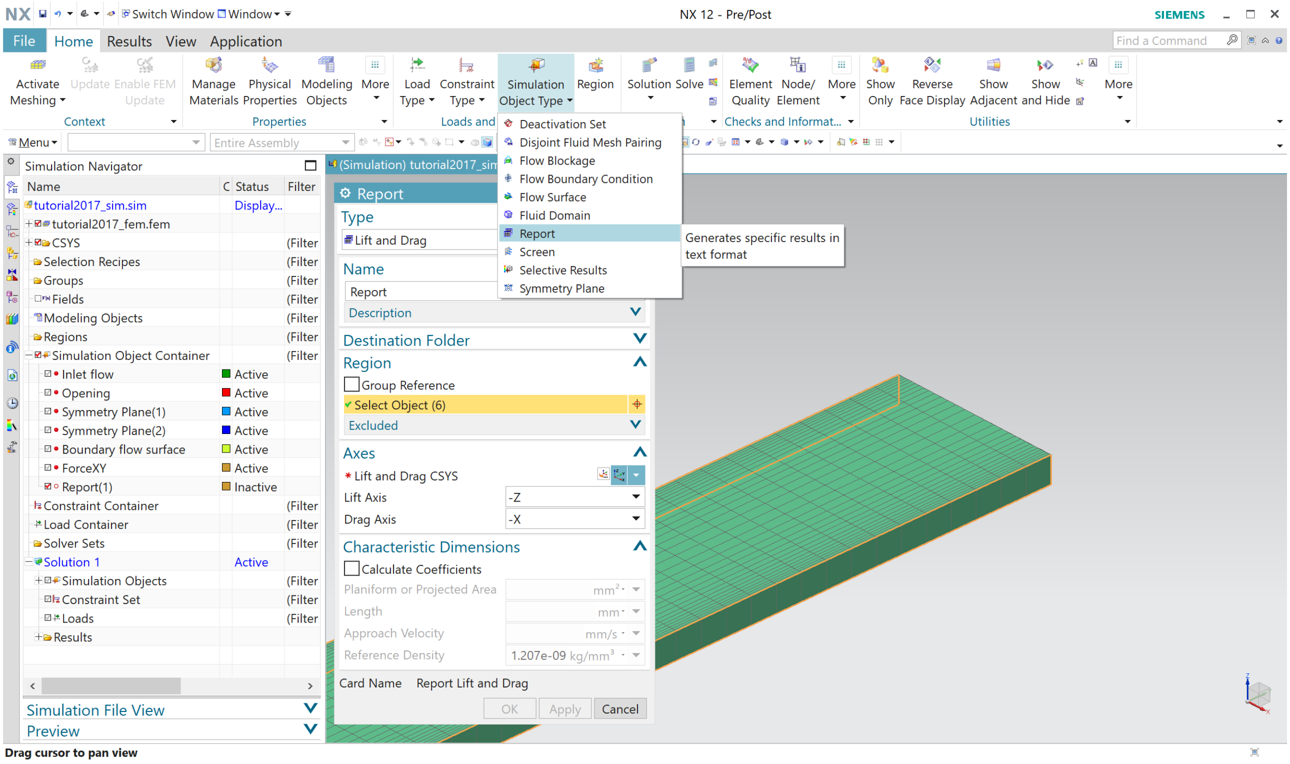
Simulation Object Type > *Report* > *Type "Lift & Drag"*. Click the surfaces of interest (i.e. the walls, thus the 6 faces along the thickness, as done in the figure below).

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.

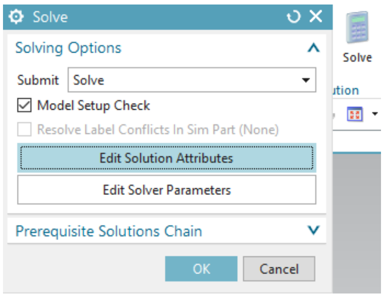
Before running the simulation, check that boundary conditions are active and that all the 3 files (`.prt`, `.fem` and `.sim`) are in the same folder.


NX 12 - Pre/Post
SIEMENS
File Home Results View Application
Simulation Object Type
Deactivation Set
Disjoint Fluid Mesh Pairing
Flow Blockage
Flow Boundary Condition
Flow Surface
Fluid Domain
Report
Screen
Selective Results
Symmetry Plane
Generates specific results in text format
Simulation Navigator
tutorial2017_sim.sim
tutorial2017_fem.fem
Simulation Object Container
Inlet flow
Opening
Symmetry Plane(1)
Symmetry Plane(2)
Boundary flow surface
ForceXY
Report(1)
Solution 1
Report
Type
Lift and Drag
Name
Report
Description
Destination Folder
Region
Group Reference
Select Object (6)
Excluded
Axes
Lift and Drag CSYS
Lift Axis -Z
Drag Axis -X
Characteristic Dimensions
Calculate Coefficients
Planiform or Projected Area
Length
Approach Velocity
Reference Density 1.207e-09 kg/mm³
Card Name Report Lift and Drag
OK Apply Cancel
Drag cursor to pan view

4 Solve the simulation

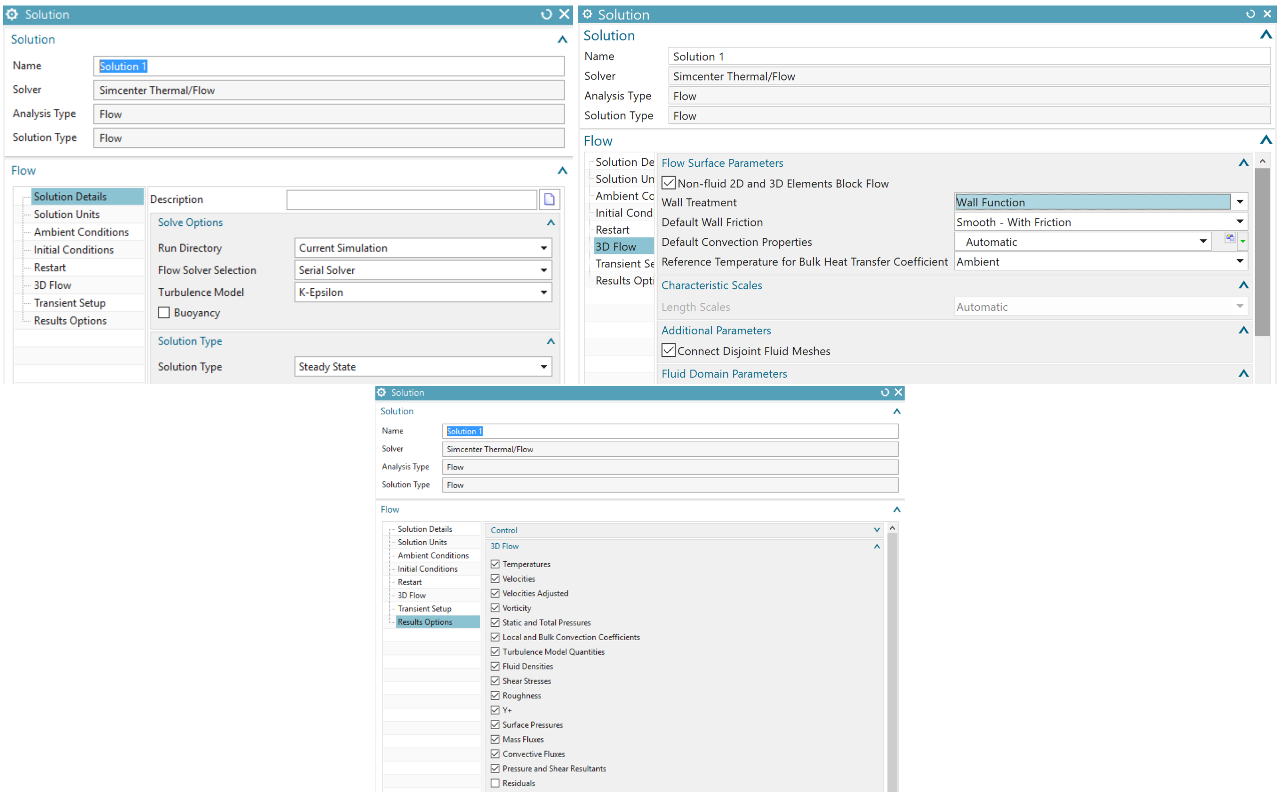
4.1 Set the solution attributes

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



Under Solution Details :

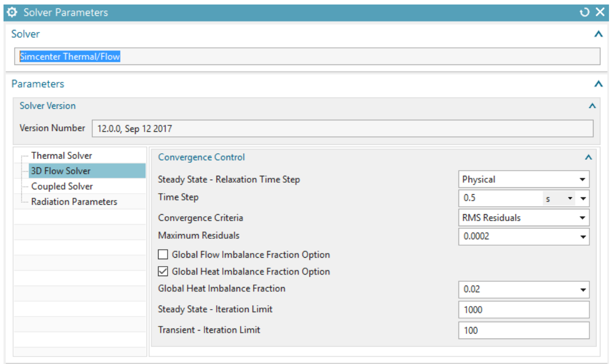
- You can select the *Solution Type “Steady state”* or *“Transient”*. In this tutorial, you should start with steady-state.
- You can select the turbulence model. The common one to be used is *"K-Epsilon"*.
- Under *3D Flow*, you should enable *Use Wall Function*.
- Under *Results Options*, you should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



4.2 Set the solver parameters

Solution > *Solve* > *Edit Solver Parameters*

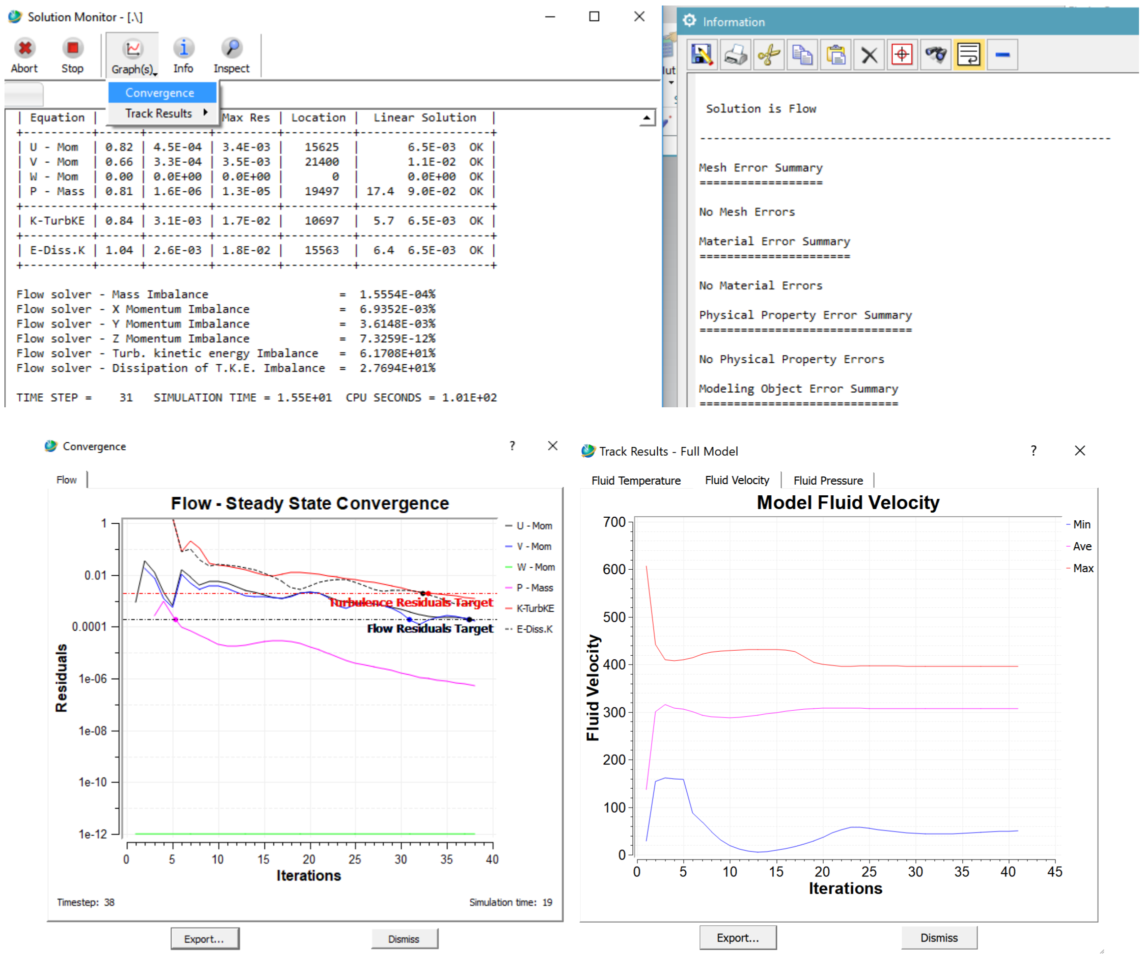
In some cases, you might need to play with the *“Steady-State Relaxation Time Step”* and the “*Time Step*” under the *3D Flow Solver* tab. It has to respect the Courant–Friedrichs–Lewy (CFL) condition $U\Delta t/\Delta x \leq 1$, with U the velocity (m/s), Δt the time step and Δx the minimal mesh size. 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.



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.

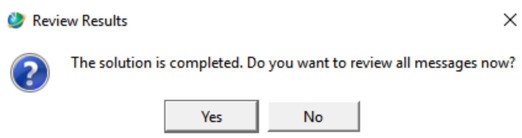


Export : you can export the graph of convergence/results in `.png` directly or in `.csv` to plot it after in Matlab.
Remark : As the simulation is purely bi-dimensional, the residual for *W* (velocity along z-axis) is constant at 10^{-12}. This is a quick check for any mistake that could make the flow 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

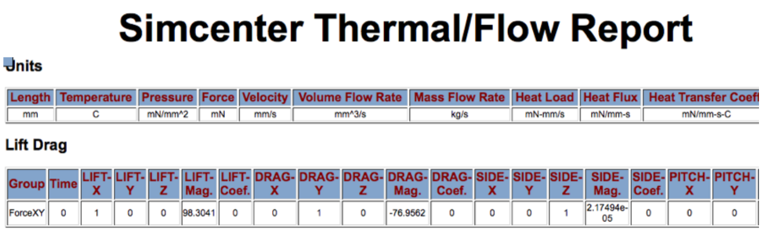
5.1 Review the verbose

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

Units

Length	Temperature	Pressure	Force	Velocity	Volume Flow Rate	Mass Flow Rate	Heat Load	Heat Flux	Heat Transfer Coeff
mm	C	mN/mm^2	mN	mm/s	mm^3/s	kg/s	mN-mm/s	mN/mm-s	mN/mm-s-C

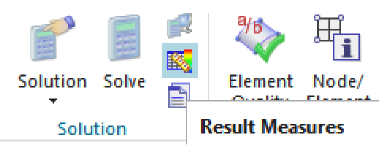
Lift Drag

Group	Time	LIFT-X	LIFT-Y	LIFT-Z	LIFT-Mag.	LIFT-Coef.	DRAG-X	DRAG-Y	DRAG-Z	DRAG-Mag.	DRAG-Coef.	SIDE-X	SIDE-Y	SIDE-Z	SIDE-Mag.	SIDE-Coef.	PITCH-X	PITCH-Y
ForceXY	0	1	0	0	98.3041	0	0	1	0	-76.9562	0	0	0	1	2.17494e-05	0	0	0

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.

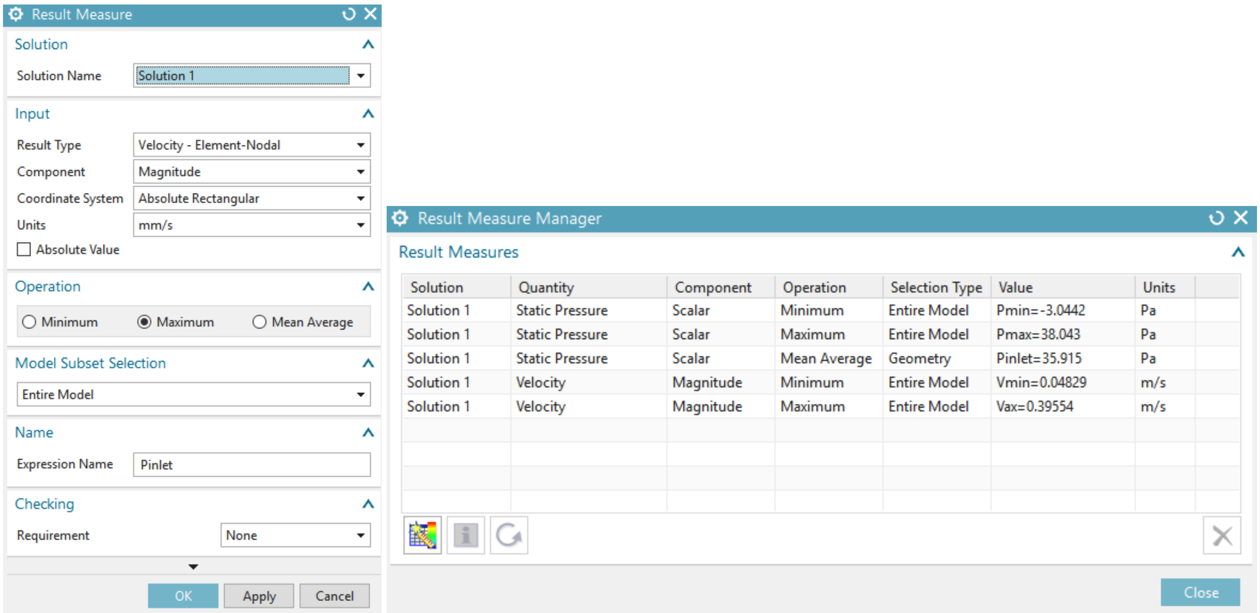
Solution tab > Result Measures



Then, click on the lower left corner NEW 

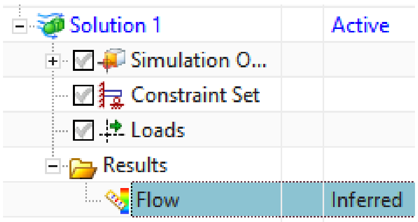
Pick up values of interest to evaluate validity and convergence of the solution : like minimum pressure over a surface or maximum velocity over the entire model.

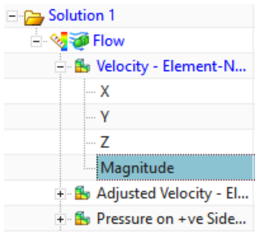
In this tutorial, we will select the min and max velocity and the min and max pressure over the entire domain, and the average pressure at the inlet.

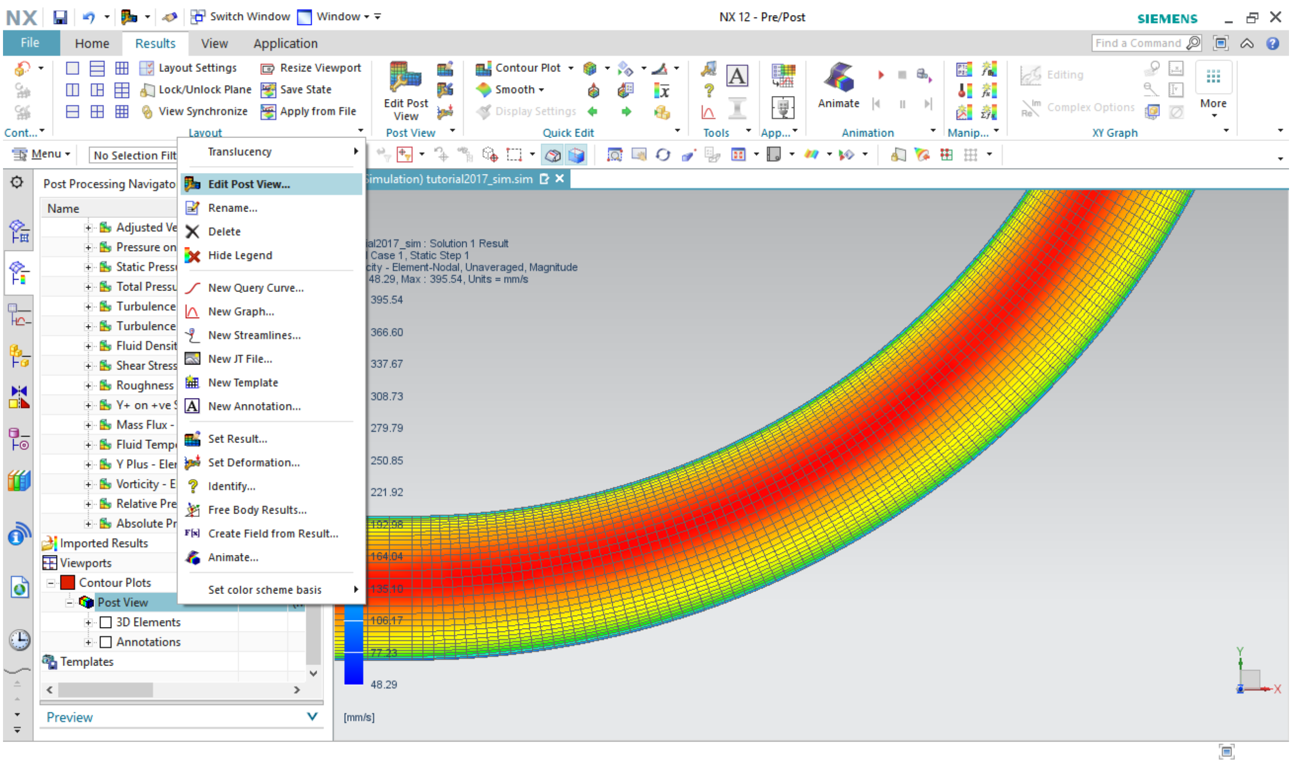


5.4 Plot the results

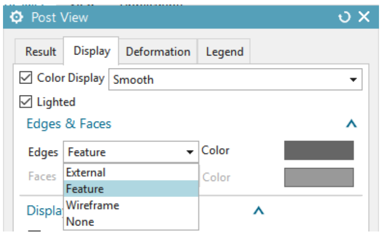
On the very bottom of the three structure, double click on *Flow* under *Results*. Then, under *Flow*, you can access the different data field : velocity, pressure,... To plot the velocity for instance, Click on *Velocity* > *Magnitude*



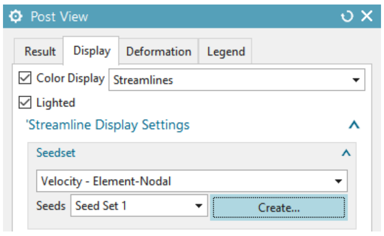


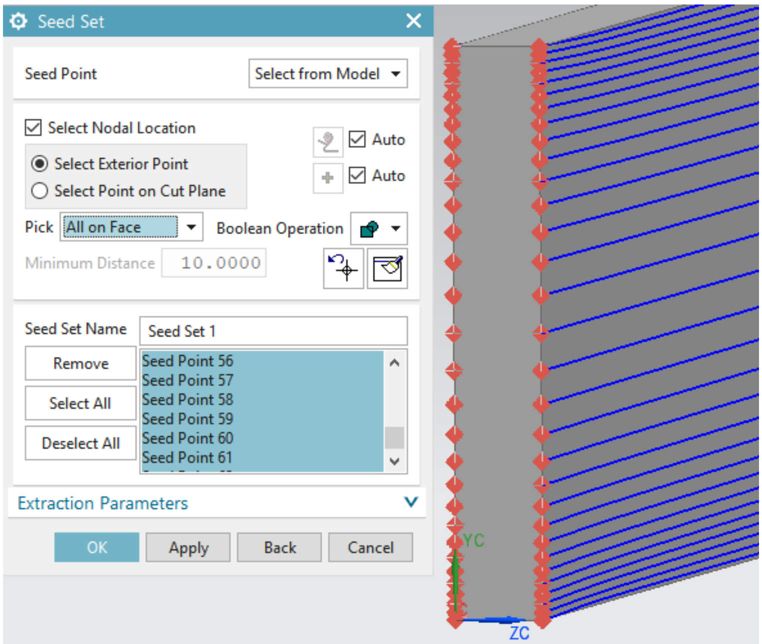


Edit Plot : At the bottom of the tree, right-click the active *Post-view* > *Edit Post View*. Or, click directly on the *Edit Post View* button on the top tool bar (*Results* tab). You can choose the *Color Display : iso-lines, arrows...*. For clarity, you can remove the display of the mesh under *Edges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

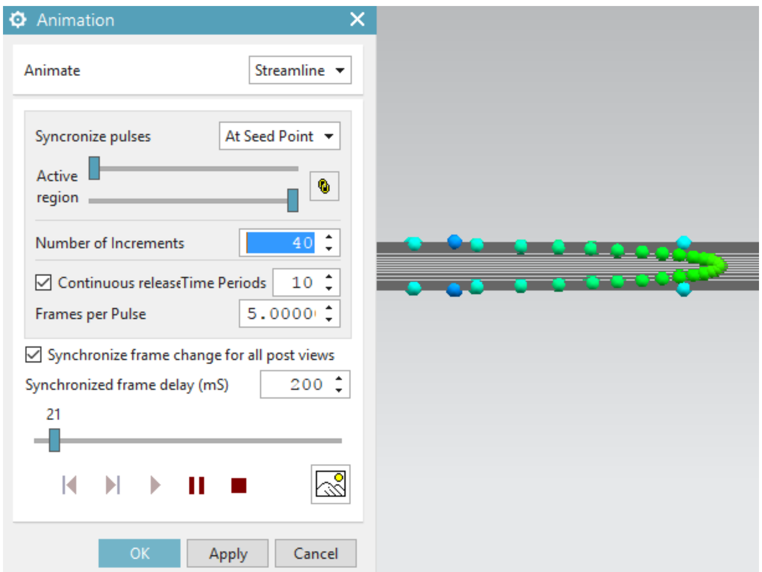


Streamlines : To plot streamlines, choose *Color Display "Streamlines"*, under *Seedset* tab, click on *Create*. On the *Seed Set* window, thick *Select Nodal Location* and *Pick "All on face"* on the inlet face for example, to plot streamlines in the whole pipe.



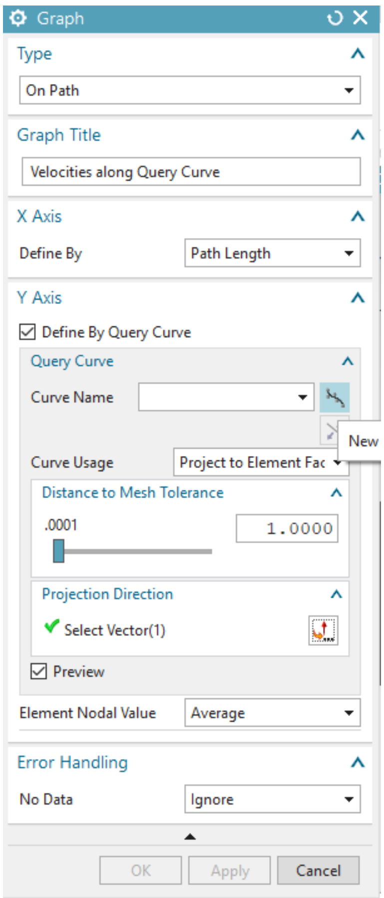


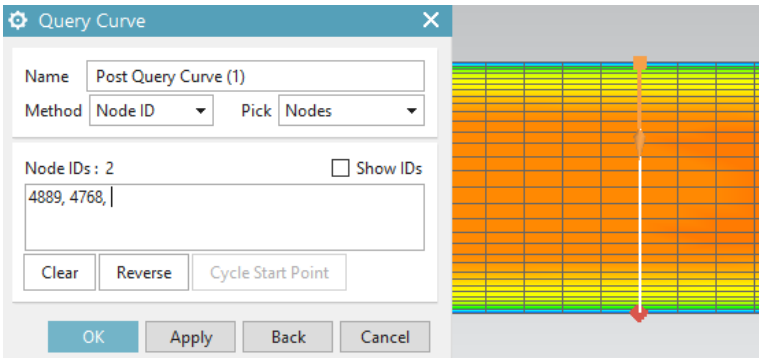
Animation : Click on *"Animate"*  to create an animation of particles on streamlines. You can export it as `.gif` file.

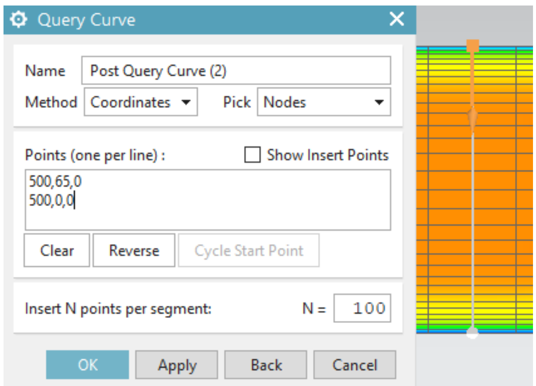


2D graph : Once you have plotted a data field (velocity magnitude for example), you can also create *2D graph* under *Tools* toolbox. In this tutorial, we will plot the velocity profile along the pipe :

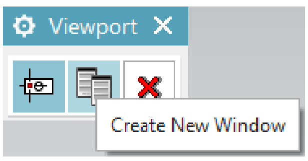
— *Tools > Create Graph*
— Under *X Axis*, *Define By "Path Length"*
— Under *Y Axis*, Tick *"Define By Query Curve"*
— Under the *Query curve* tab, click the top right icon to create a new curve.
— Give a name to the curve and select
  — *Method "Nodes ID"* and pick up "Nodes" with the pen (by selecting 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.



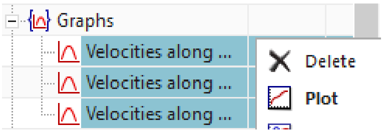




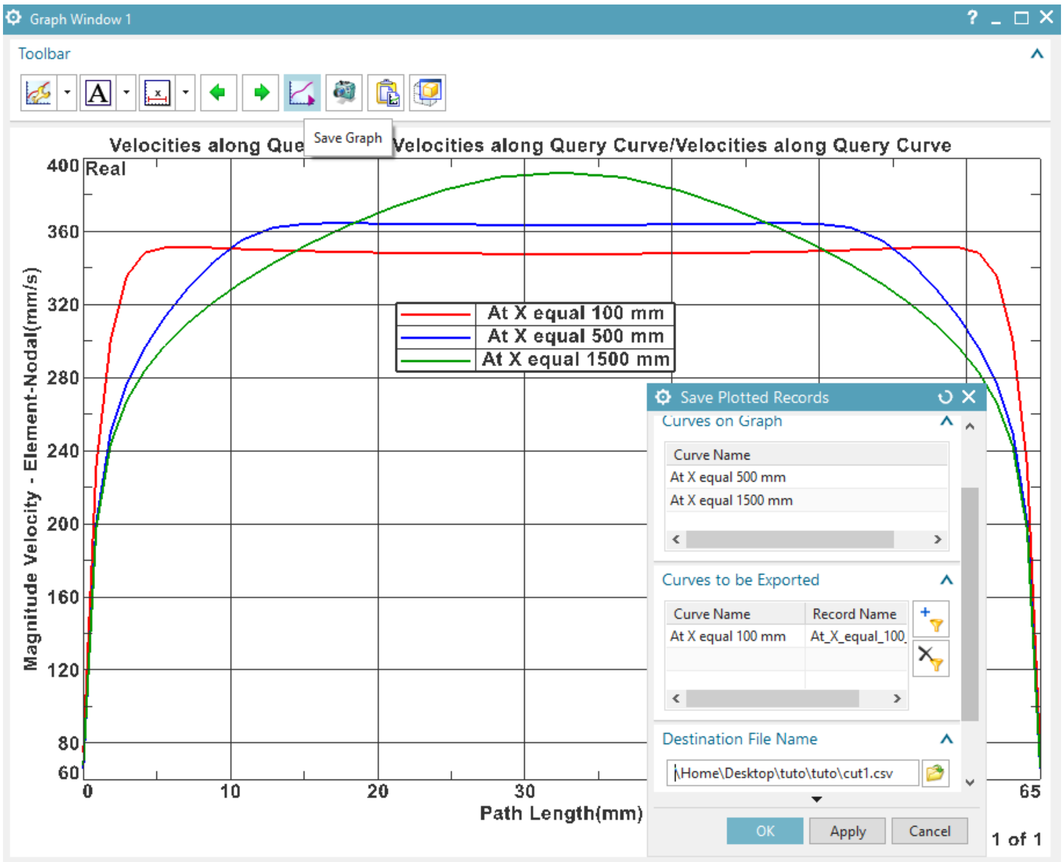
After *OK*, a small pop-up window will then ask you if you want to plot it in a new window or in an existing one, chose to *Create New Window*.



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*.



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.

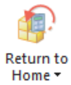


Export 2D graph : On a *Graph Window*, cllick on *Toolbar* and then *Save Graph*. When you have several plots, choose which one to *Export* and select `.csv` as file format (to import and show in Matlab for example).

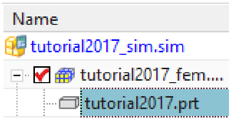
A tutorial on how to plot the solution is available here (steps 6 to 9) :
https://docs.plm.automation.siemens.com/data_services/resources/nx/12/nx_help/training/en_US/advanced_sim_tutorial/id741086.html

6 Change the design

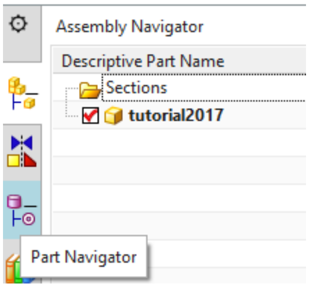
In this paragraph, we will show how to test an alternative design, for instance a shorter radius corner.

- First click the top-left icon *Return to home* 

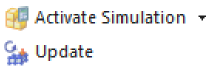
 to exit the result analysis and go back to the simulation navigator.
- In the tree structure, under `yourproject_sim.sim/yourproject_fem.fem` double-click `yourpart.prt` two times in a row to open it again. Now you can switch between the part, the mesh and the simulation via the button in the header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



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

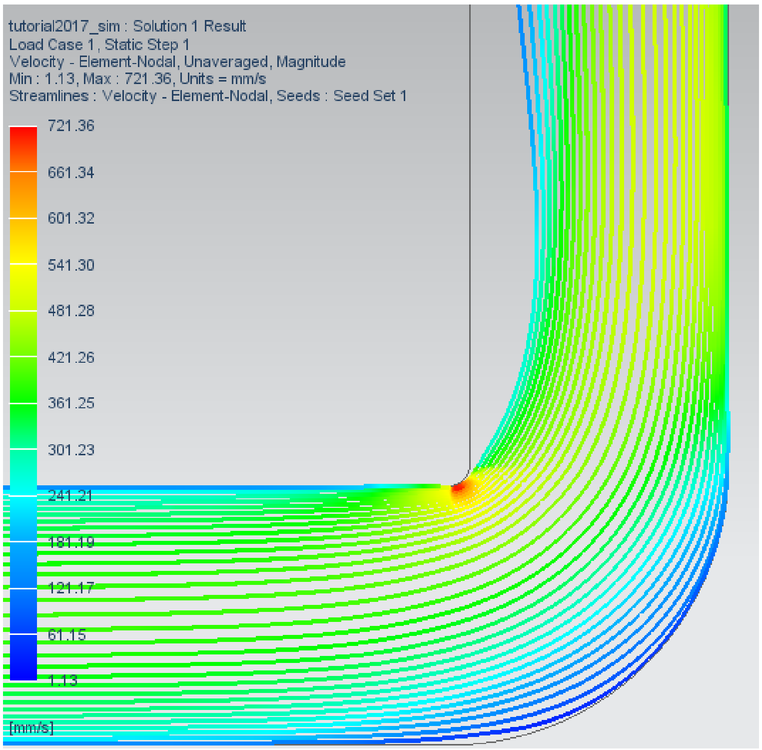


- Now you can edit your sketch by double-clicking it. Change the radius from 400 to 70 mm.
- Click *Finish Sketch*.
- *Switch window* to go back to the *Simulation*.
- In the top-left, click *Activate Meshing*

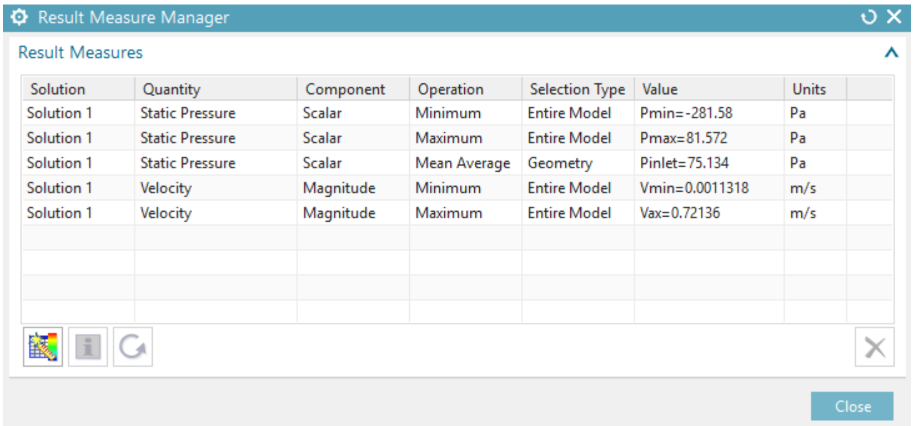
- In the top-left, click *Update*, to automatically update the meshing to the new geometry. Then again in the top left *Activate the Simulation* 

 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.



	$R = 400$ 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

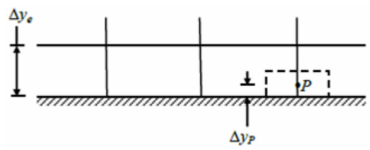
As expected, you can see that the inlet pressure is higher, more than twice what was needed for the large radius. Minimum, maximum pressure and maximum velocity are also higher in absolute values over the domain.

Compared to the analytical prediction ($\Delta p = 31.2$ 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.

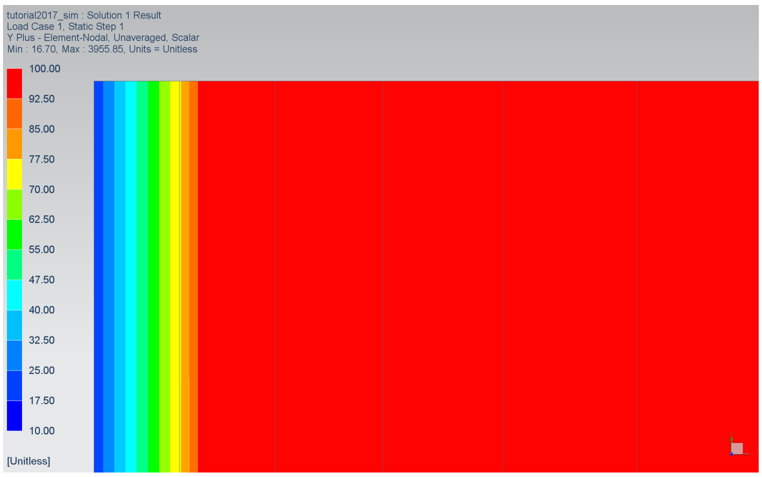
7 Change the turbulence model or solution type

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).



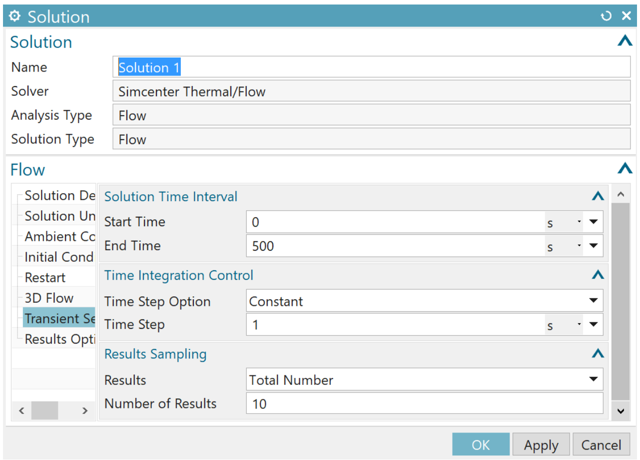
This is computed by selecting *Y+* and *Turbulence Model Quantities* (containing *Y Plus*) in *Results Options* before solving the simulation. The validity of y^+ is crucial since the boundary layer has to be well modeled. You can check it by displaying *Y Plus* in the results. Then, instead of *"smooth"* 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

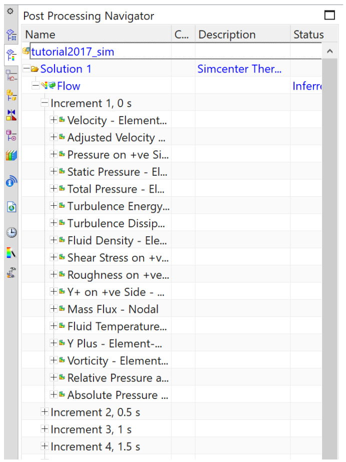


7.2 Solution type

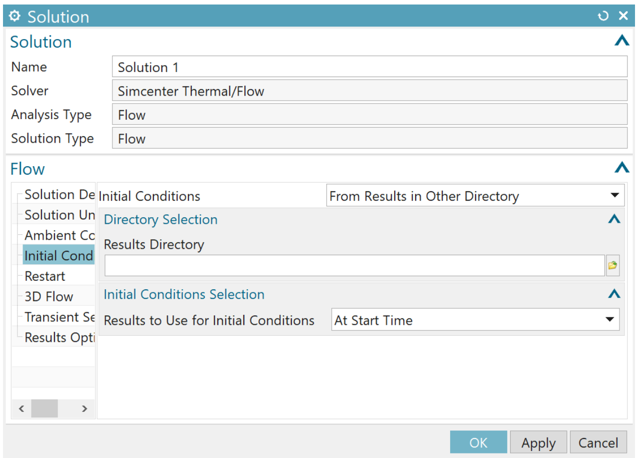
Depending on the problem, you might solve a transient (unsteady) flow. This can be chosen in *Solve* > *Edit Solution Attributes* > *Solution Details* > *Solution Type* > *"Transient"*. In this case, you have to select the *Transient Setup* :

- *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.

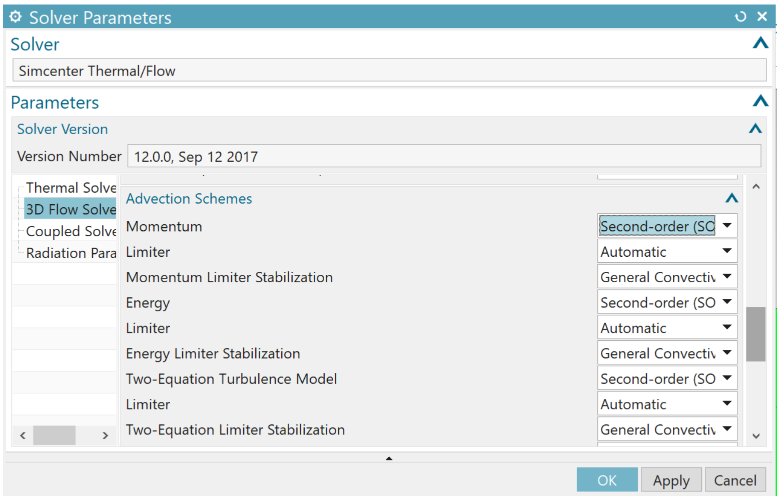




It is also possible to add *Initial Conditions* from a specified *"Uniform"* state or *"From Results in Other Directory"*, to start from a previous state, which is already computed.

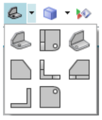


To trigger unsteady phenomena (for example, to force vortex shedding behind a cylinder), you can change the solver scheme : go to *Solve > Edit Solver Parameters > 3D Flow Solver > Advection Schemes* and change *"First-order"* to *"Second-order (SOU)"*.



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.



— **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