Using Siemens NX 11 Software

Advanced Finite Element Analysis - Valve

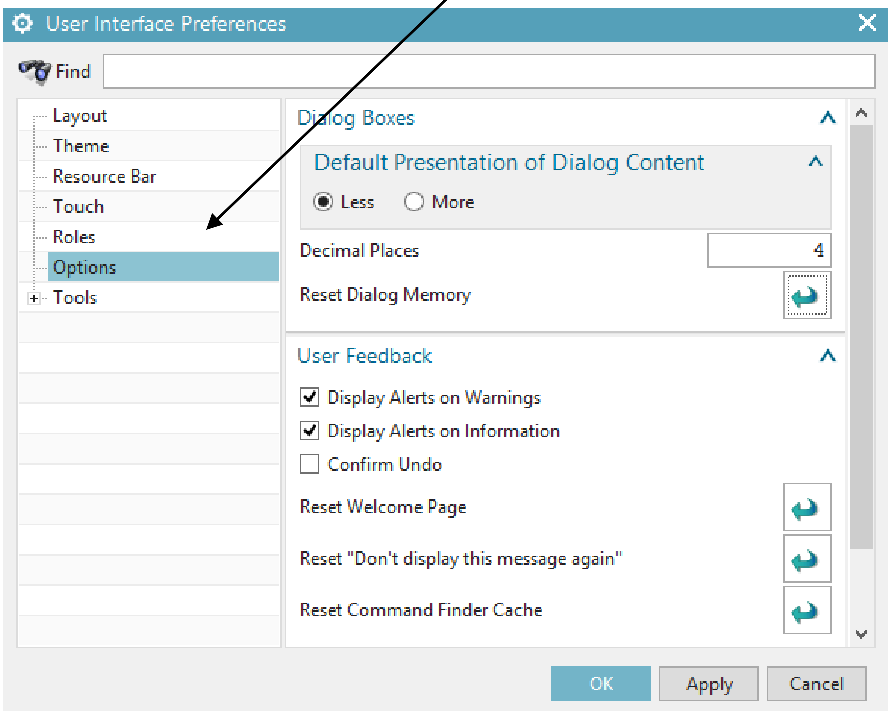
1 – Introduction.

The goal of this tutorial is to perform a finite element analysis (FEA) of a water flow running through a pipe which is heated.

- Make a **copy** of the file *C:\Commun\NX \fem\valve.prt* file in your local folder, and open it.

2 – Reset dialog boxes.

- Click on *Menu->Preferences->User Interface...*
- In the *User Interface Preferences* dialog box, click on the *Options* item of the left tree, and then on the *Reset Dialog Memory*  button.
- Click *OK* to validate.

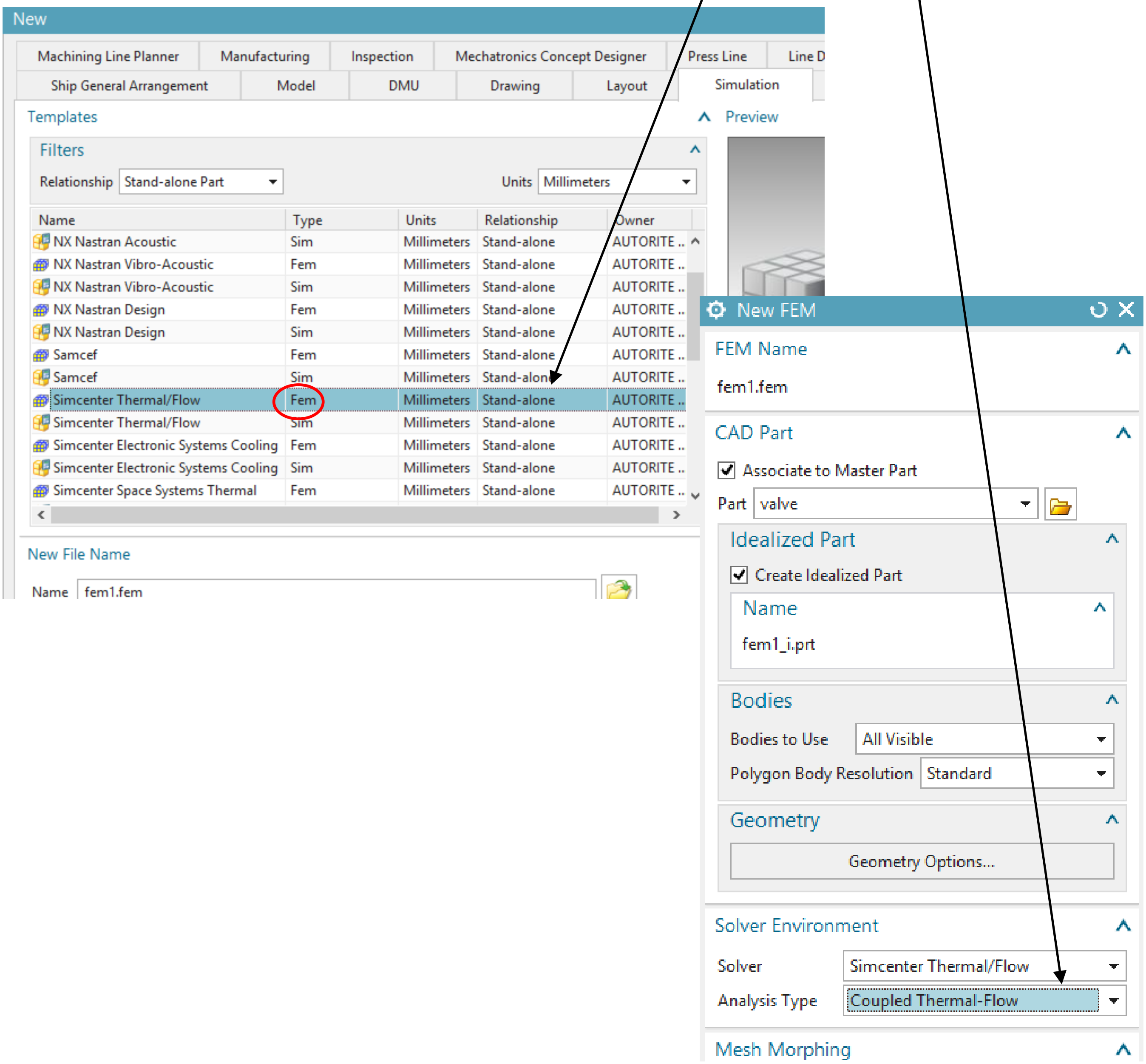


In what follows, we will perform the next couples of steps

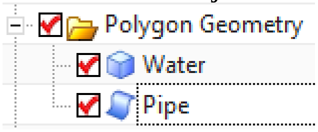
1. Creating meshes (2D and 3D) and their associated material properties for defining how the water and the pipe will behave under geometric and thermal constraints.
2. Applying boundary conditions, loads and couplings.
3. Computing the FEA.
4. Visualizing the results.

3 – Creating a new FEM file.

- Create a new FEM file by clicking on *File->New* and selecting under the *Simulation* tab of the *New* dialog box, the item *Simcenter Thermal Flow* ***(FEM)***.
- In the *New FEM* dialog box, set the option *Analysis Type* of the *Solver Environment* field to **Coupled Thermal-Flow**.



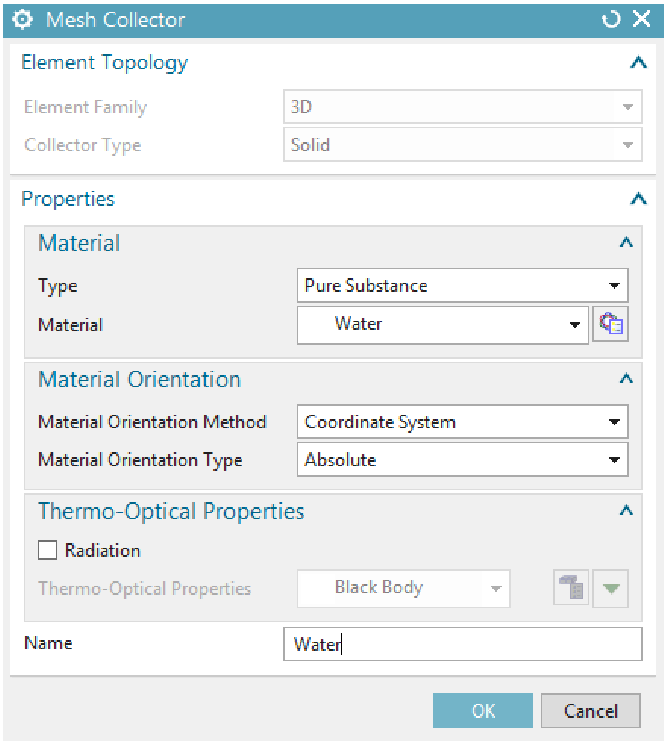
4 – Meshes and associated material properties.

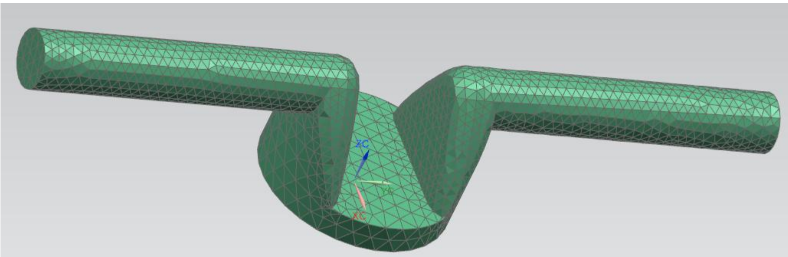
- In the tree of the *Simulation Navigator* , click on the + symbol in front of the *Polygon Geometry* item Polygon Geometry.
- You can see two *Polygon Body* objects. Rename the first one *Water* (click-right on it->*Rename*) and the second one *Pipe*. The first object will model the fluid (water), while the second object will model the pipe through which the water is running.


- You can hide/show these objects by clicking on their respective checkboxes.

- Select the *Pipe* and assign to it the material Assign Materials *Aluminium_2014*.
- Select the *Water* and assign to it the material Assign Materials *Water*.
- Select the *Water* and click on the 

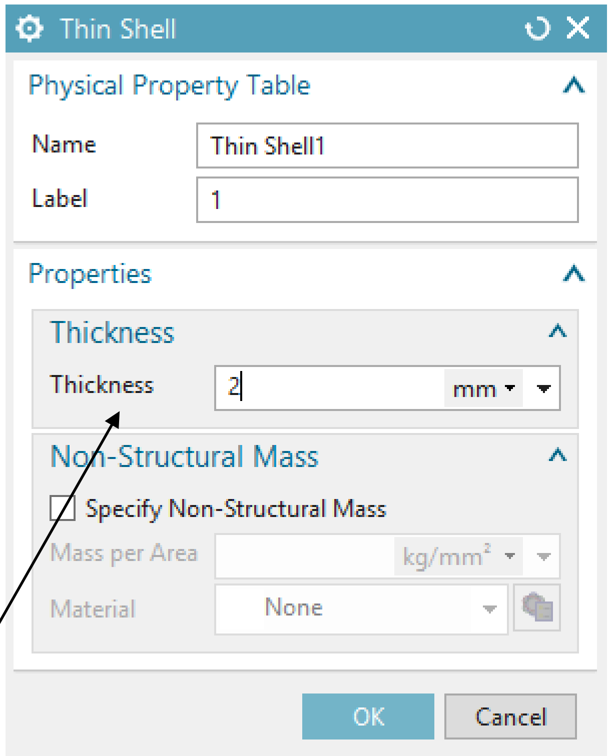
3D-Tetrahedral button. In the *3D tetrahedral Mesh* dialog box, select the **TET4** mesh type with an element size of **10 mm**.
- In the *Destination Collector* field click on the *New Collector*  button. A *collector* associates a mesh with a material.
- In the *Mesh Collector* dialog box, set the *Name* option to **Water**, choose **Water** as material, and uncheck the *Radiation* checkbox.
- Click *OK* to return to the previous dialog box, and then click *OK* again.
- The 3D mesh associated to the material properties of water should be created.



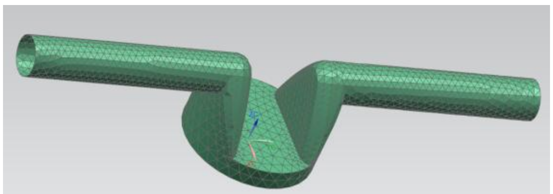


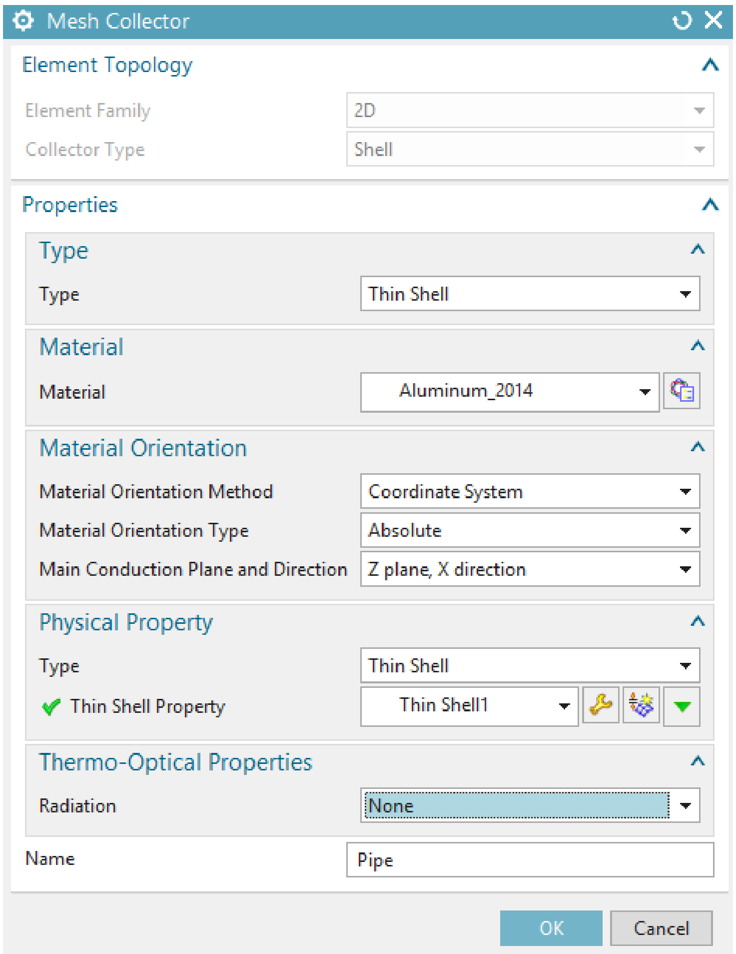
- Select the *Pipe* and click on the *2D Mesh*

button.
- In the *2D Mesh* dialog box, select the **TRI3 Thin Shell** element type with an element size of **10 mm**.
- In the *Destination Collector* field, click on the *New Collector*  button.
- In the *Mesh Collector* dialog box, select **Aluminium_2014** as *Material*.
- In the *Physical Property* field, click on the *Create Physical*  button. This will set the behavior of the thin shell along the third dimension.
- In the *Thin Shell* dialog box, go to the *Thickness* field and set the *Thickness* option to **2 mm** (thickness of the pipe).
- Click *OK* to return to the *Mesh Collector* dialog box.



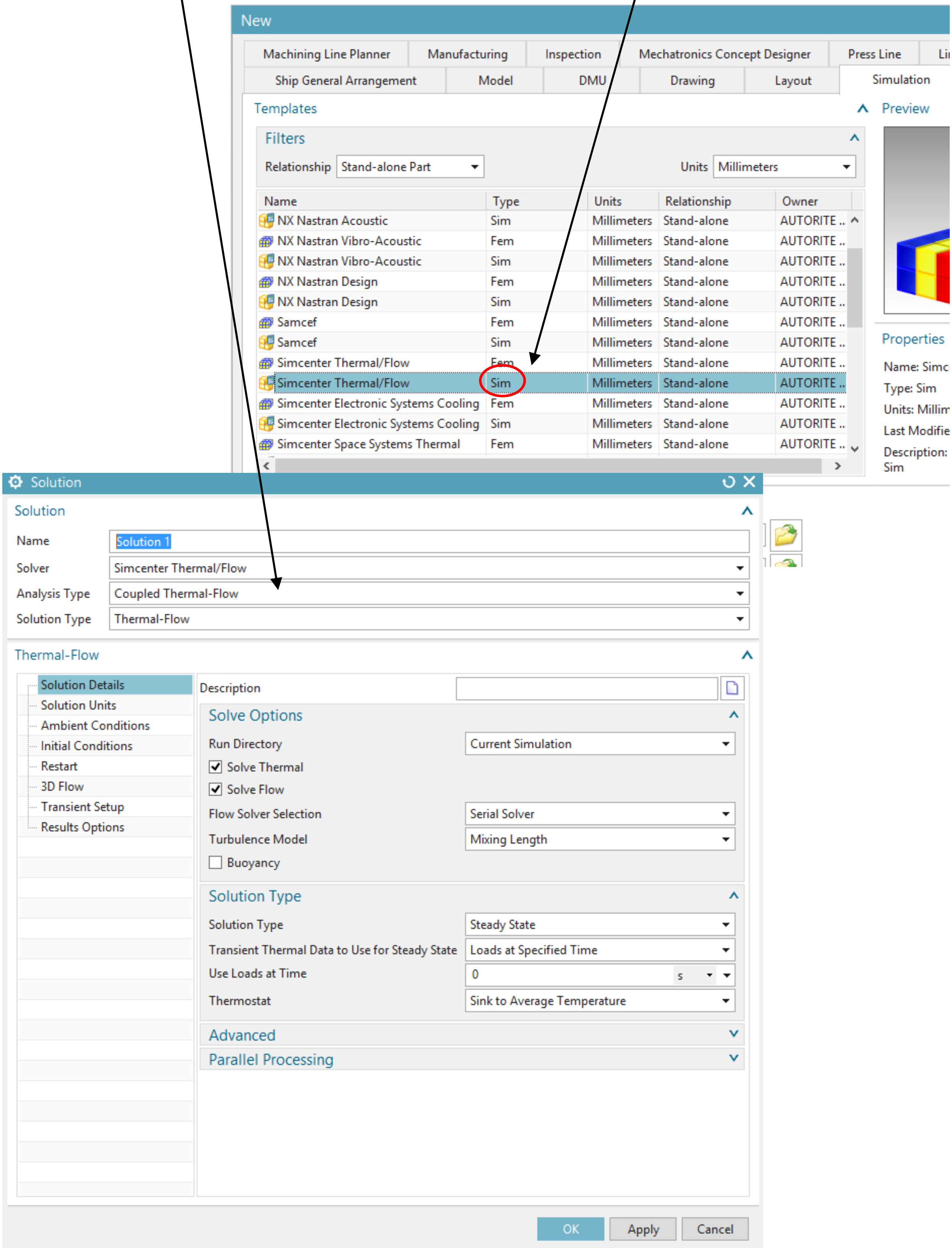
- In the *Thermo-Optical Properties* field, set the *Radiation* option to **None**.
- Set the *Name* option to **Pipe**.
- Click *OK* to return to the *2D Mesh* dialog box, and then click *OK* again.
- A 2D mesh associated to the material properties of aluminium (thin shell) should be created.
- **Save your files (ctrl+s)**.





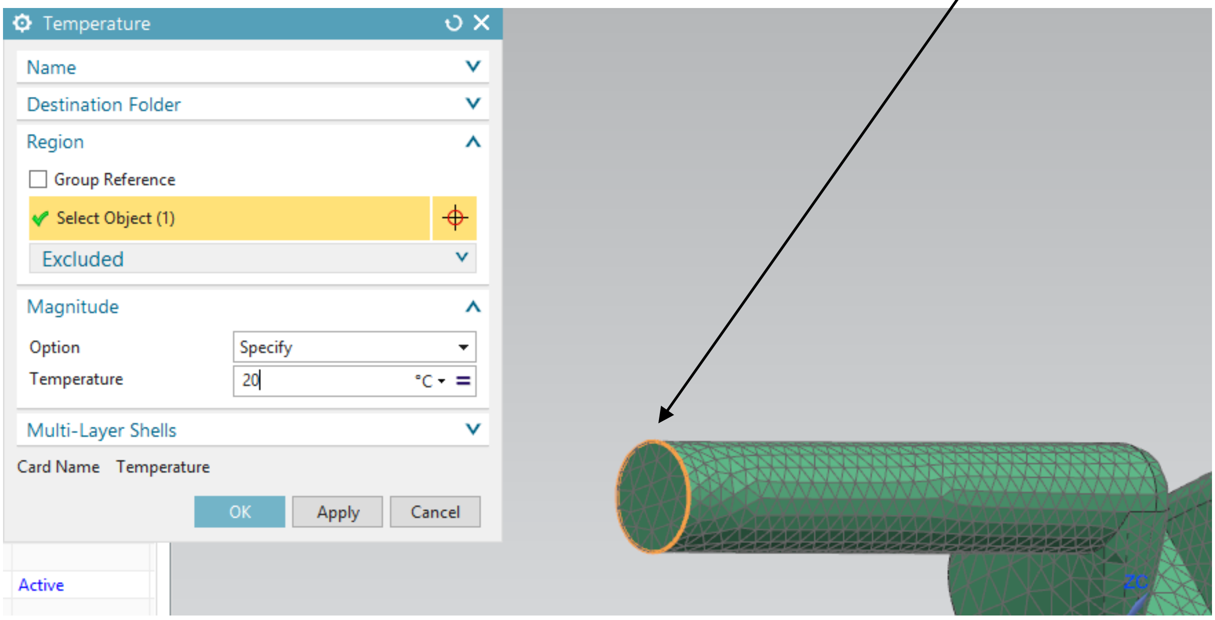
Boundary conditions, loads and couplings.

- Click on *File->New* and select *Simcenter Thermal/Flow* ***(Sim)***.
- In the *New Simulation* dialog box, simply keep the default values and click *OK*.
- In the *Solution* dialog box, select **Coupled Thermal-Flow** as *Analysis Type* and click *OK*.



5 – Boundary conditions.

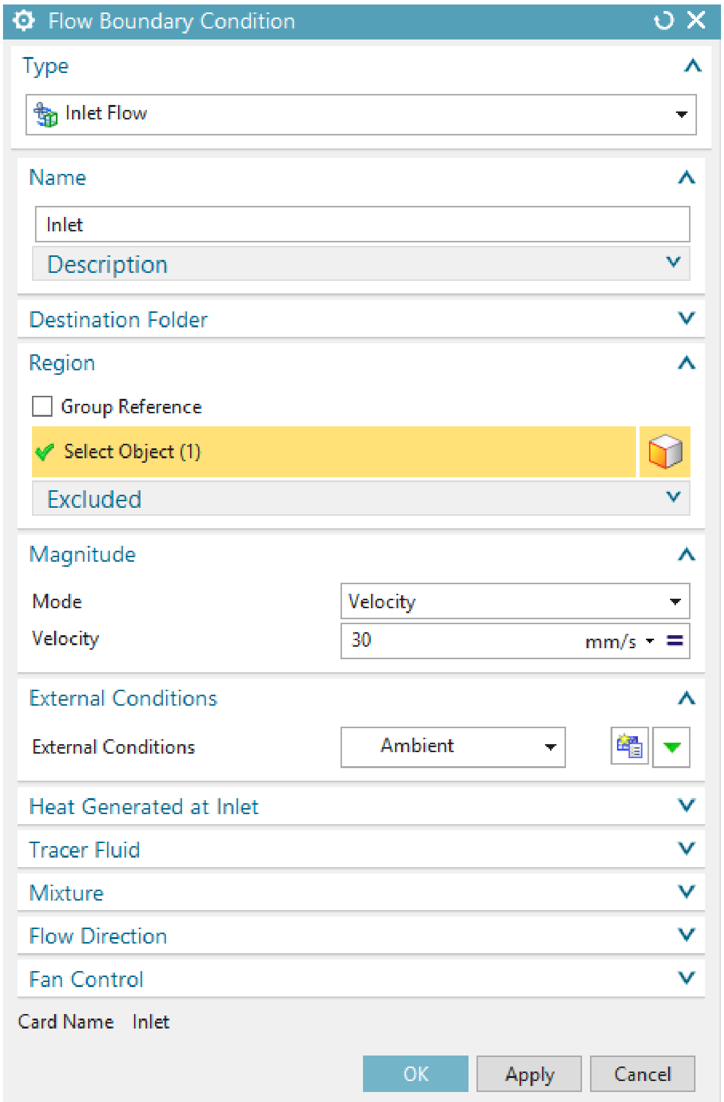
- In the *Simulation Navigator* , expand (+) the *fem1.fem* item fem1.fem . This item contains the geometry, meshes and associated materials (via the *Collectors*).
- Hide the *Pipe* polygon and its associated 2D collector (uncheck the associated checkboxes).
- In the *Loads and Conditions* group, click on the *Constraint Type* Constraint Type button and add a *Temperature* constraint Temperature .
- In the *Temperature* dialog box, specify a temperature of 20°C and select the leftmost circular face. This will set the input water in the pipe to be at a temperature of 20°C. The water will later on be heated as it passes through the pipe.
- Click *OK* to validate the constraint.



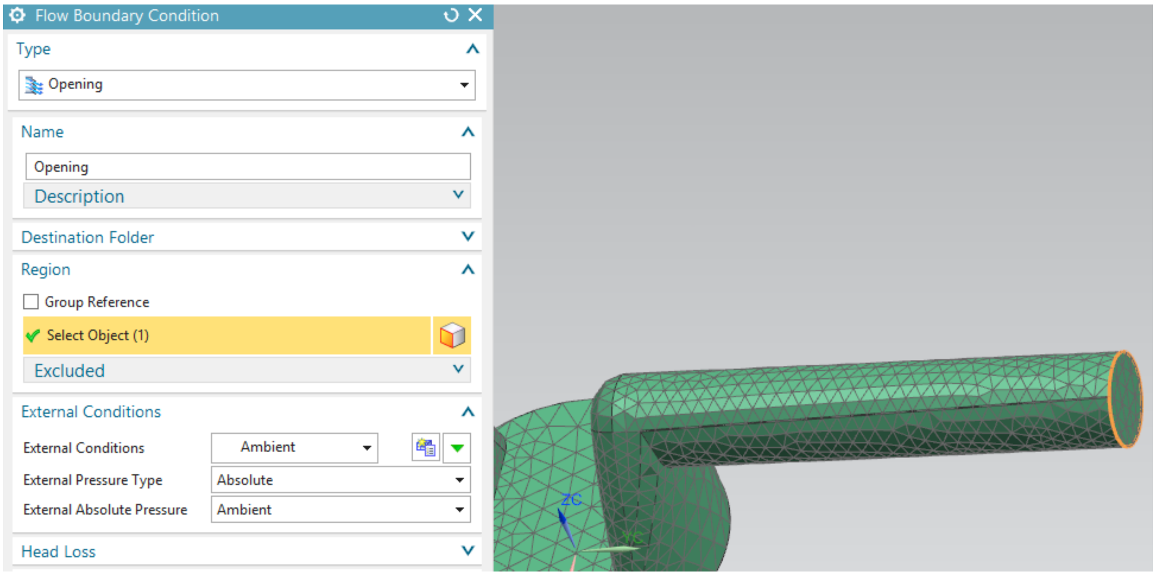
- In the *Loads and Conditions* group, click on the *Simulation Object* 

Type button, and then on *Flow Boundary Condition* 

.
- In the *Flow Boundary Condition* dialog box, select **Inlet Flow** as *Type*, and set the *Name* option to **Inlet**.
- Select as object the same leftmost circular face as previously selected, and set the *Velocity* option of the *Magnitude* field to **30 mm/s**.
- Click *Apply* to validate (keep the dialog box open for the next boundary condition).

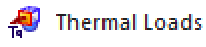


- Now, still in the same dialog box, select **Opening** as *Type* and set the *Name* option to **Opening**.
- Select the rightmost circular face, opposite to the previous leftmost circular face and click *OK*.



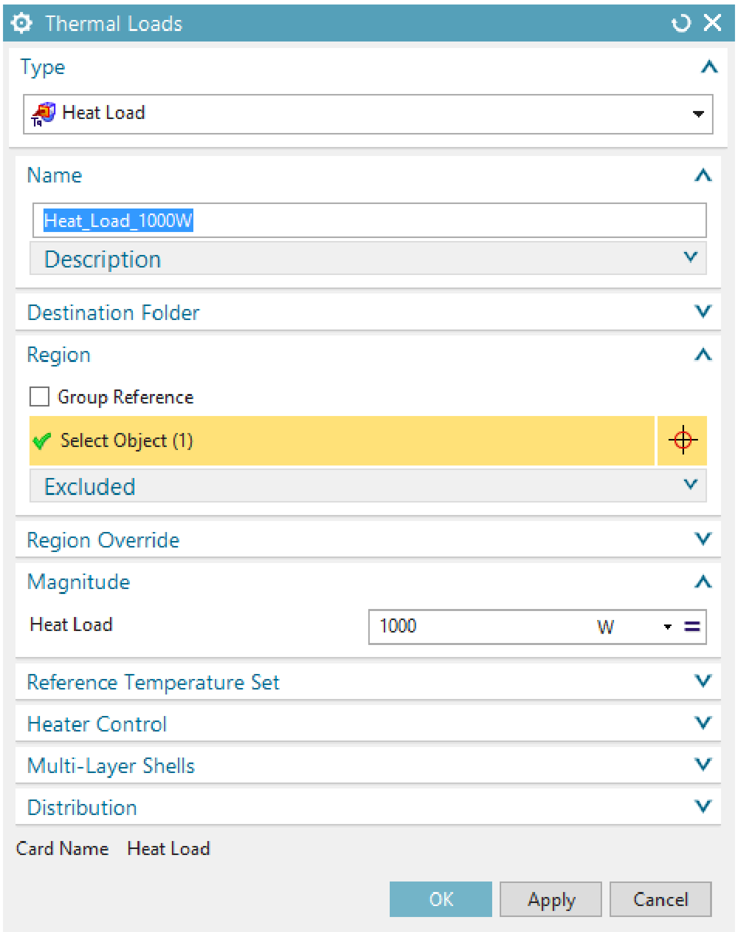
6 – Loads.

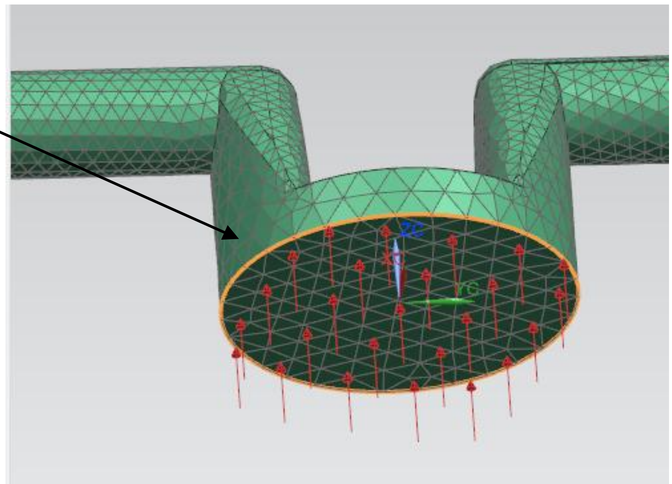
- Hide the *Water* and its associated 3D collector and show the *Pipe* and its associated 2D collector.
- In the *Loads and Conditions* group, click on the *Load Type*

button, and then on *Thermal Loads* 

.

- In the *Thermal Loads* dialog box, select **Heat Load** as *Type*.
- Set the *Name* option to **Head_Load_1000W**.
- In the *Magnitude* field, set the option *Heat Load* to **1000 W**. This will model the heat source that is heating the water through the pipe (a coupling pipe-water will be needed in order to “transport” the received heat from the metal of the pipe to the water. We will implement this coupling later on).
- As object, select the bottom most circular face of the *Pipe* and click *OK*.

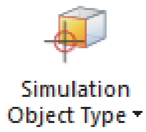




7 – Couplings.

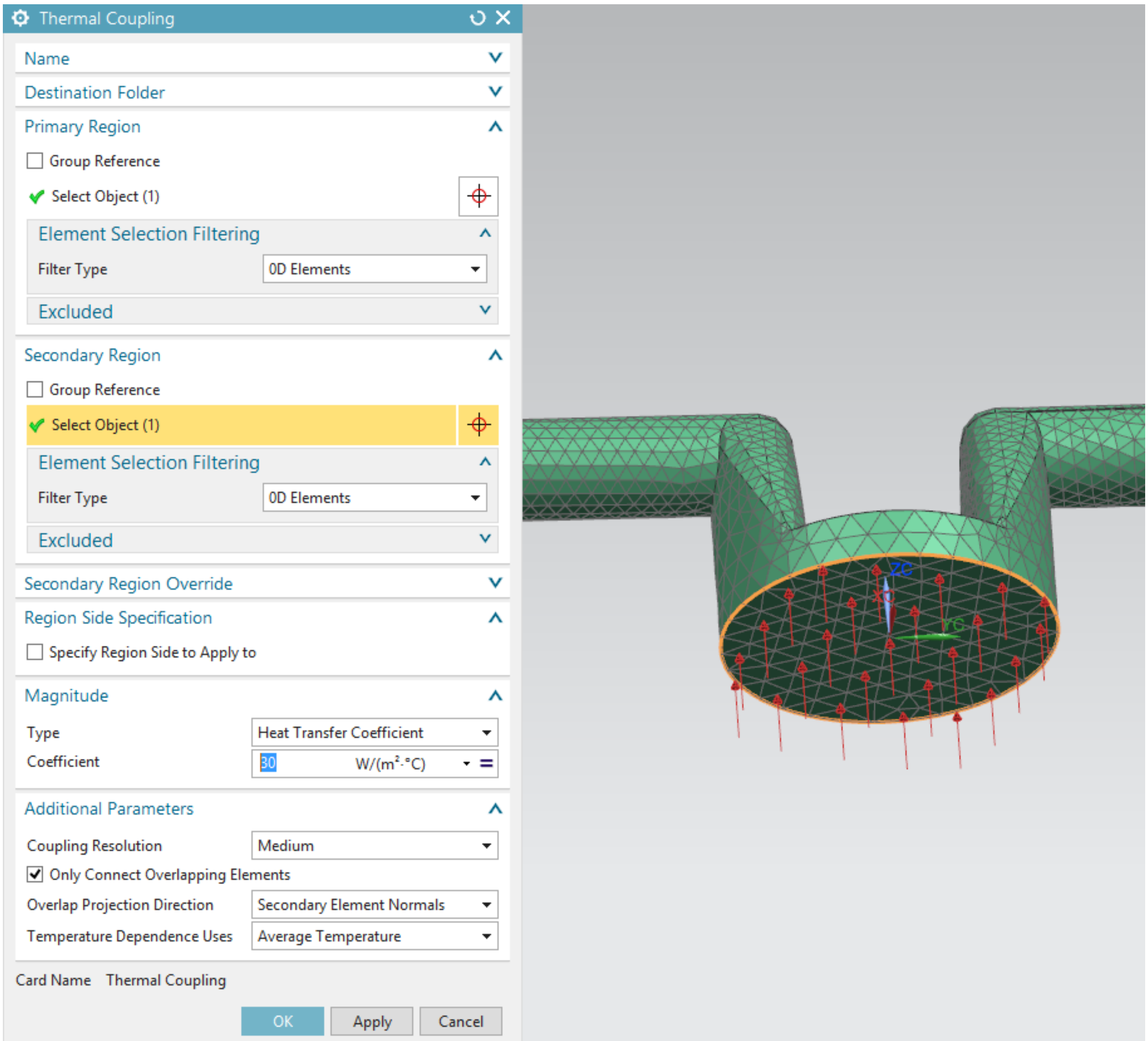
We will now couple the 2D mesh of the *Pipe* with the 3D mesh of the *Water*. This will allow the heat of the *Pipe* to propagate into the *Water*.

From now, you will have to hide and show alternatively the *Pipe* and the *Water* in order to select the correct faces.

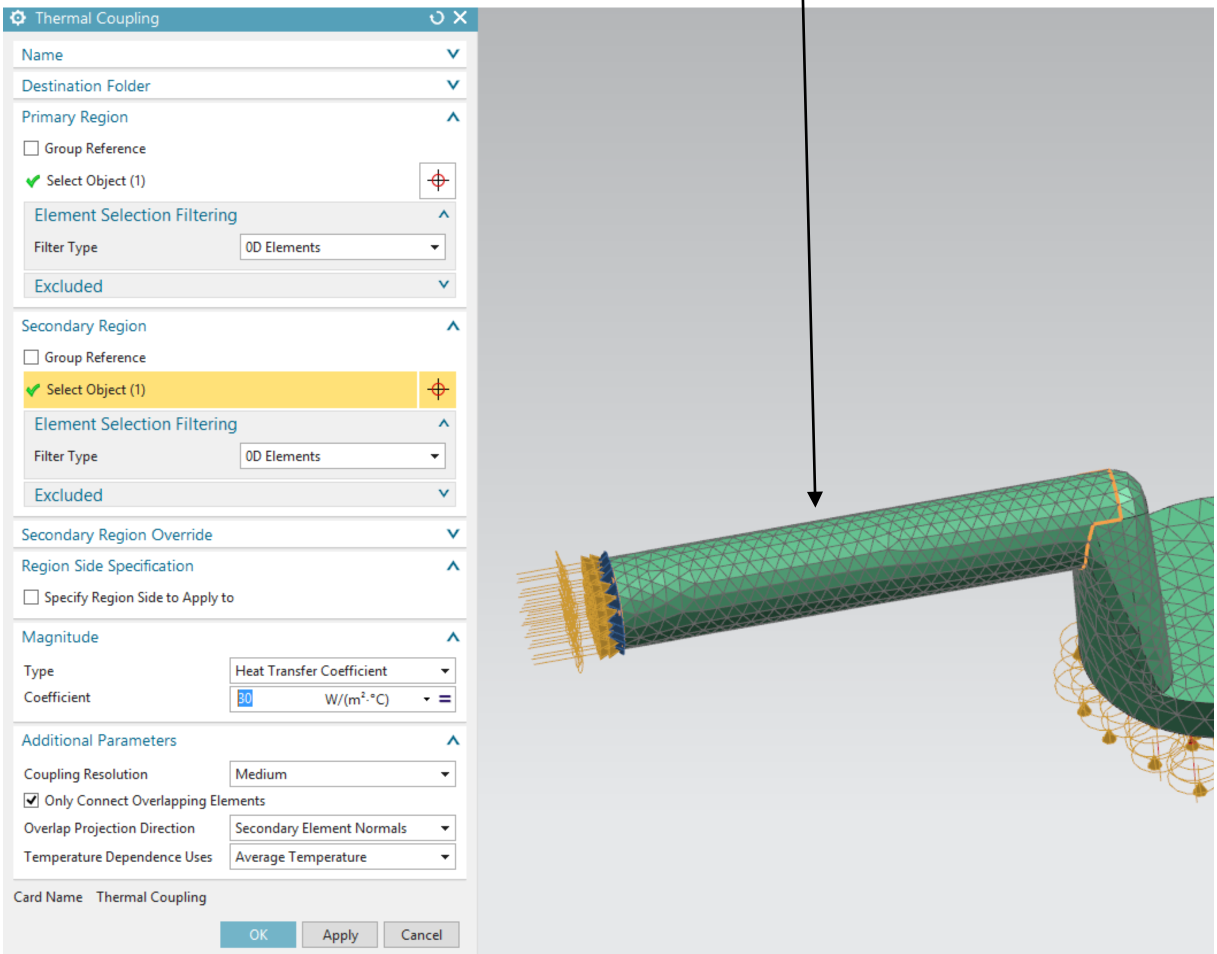


- In the *Loads and Conditions* group, click on the *Simulation Object Type* button, and then on *Thermal Coupling* 

.
- In the *Thermal Coupling* dialog box, go to the *Magnitude* field and set the *Type* option to **Heat Transfer Coefficient**. Set the *Coefficient* option to **30 W/(m² °C)**.
- Select as *Primary Region* the bottom most circular face of the ***Pipe*** (as previously done) and as *Secondary Region* the bottom most circular face of the ***Water***.
- Click *Apply* (do not close the dialog box).



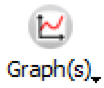
- In the same dialog box, keep the same parameters. This time, use the left cylinder surface of the ***Pipe*** as *Primary Region* and the left cylinder surface of the ***Water*** as *Secondary Region*. This coupling will allow setting the pipe to the same temperature as the water, as the later enters in the first.
- **Save your simulation files.**



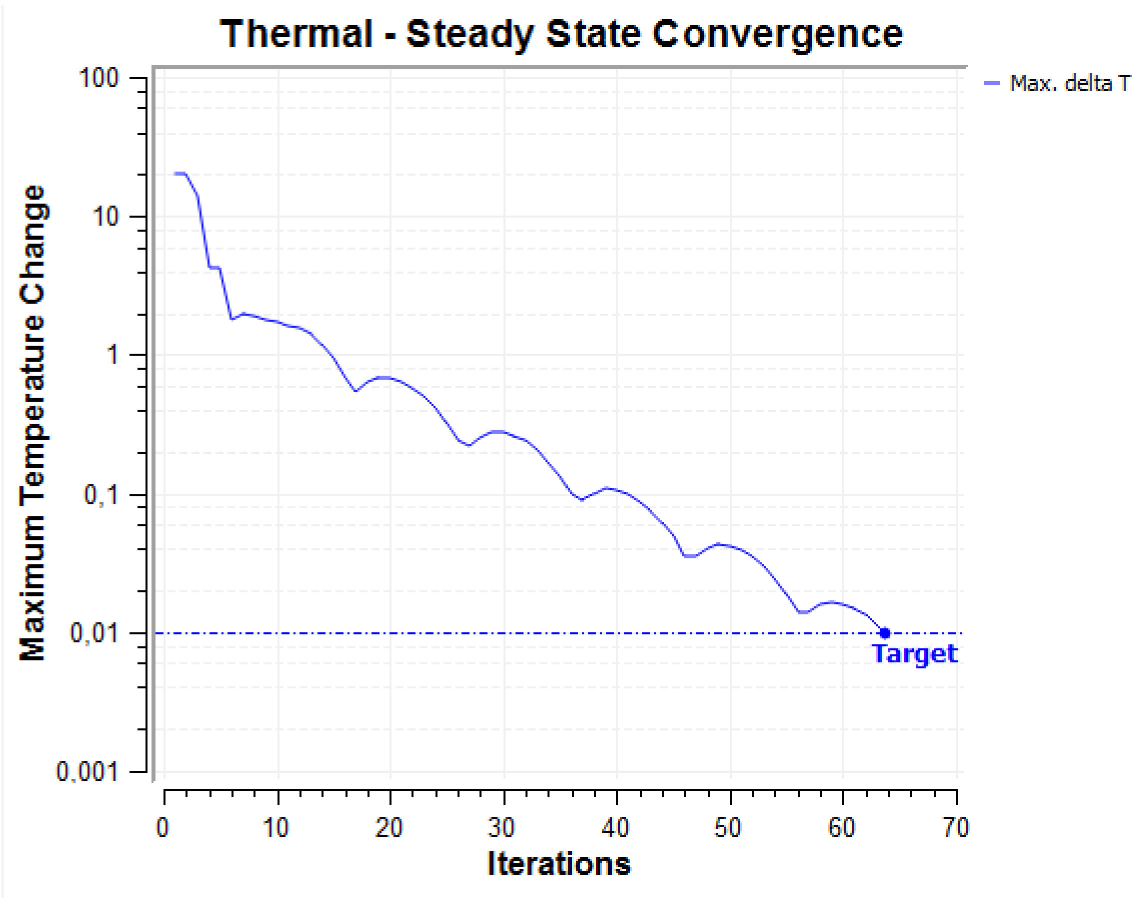
8 – Solving the model.

Now, as all physical parameters (geometry, meshes, associated materials, loads, couplings and boundary conditions) are set, we will solve the model and then visualize the different results.

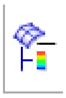
- Click on the *Solve* 

button. Keep the default values in the *Solve* dialog box and simply click *OK*. The computations should take about a minute.
- When the *Review Result* dialog box shows up, click *yes*. In the *Solution Monitor* dialog box, click on the *Graph(s) Convergence* 

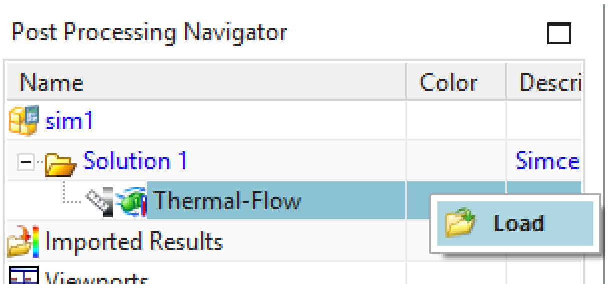
button to check if the computation indeed converge.
- You should get a convergence graph similar at the one shown here below.
- Close the graph window, the *Solution Monitor*, the *information Window* and the *Analysis Job Monitor* (*Cancel*).



Visualizing the results.

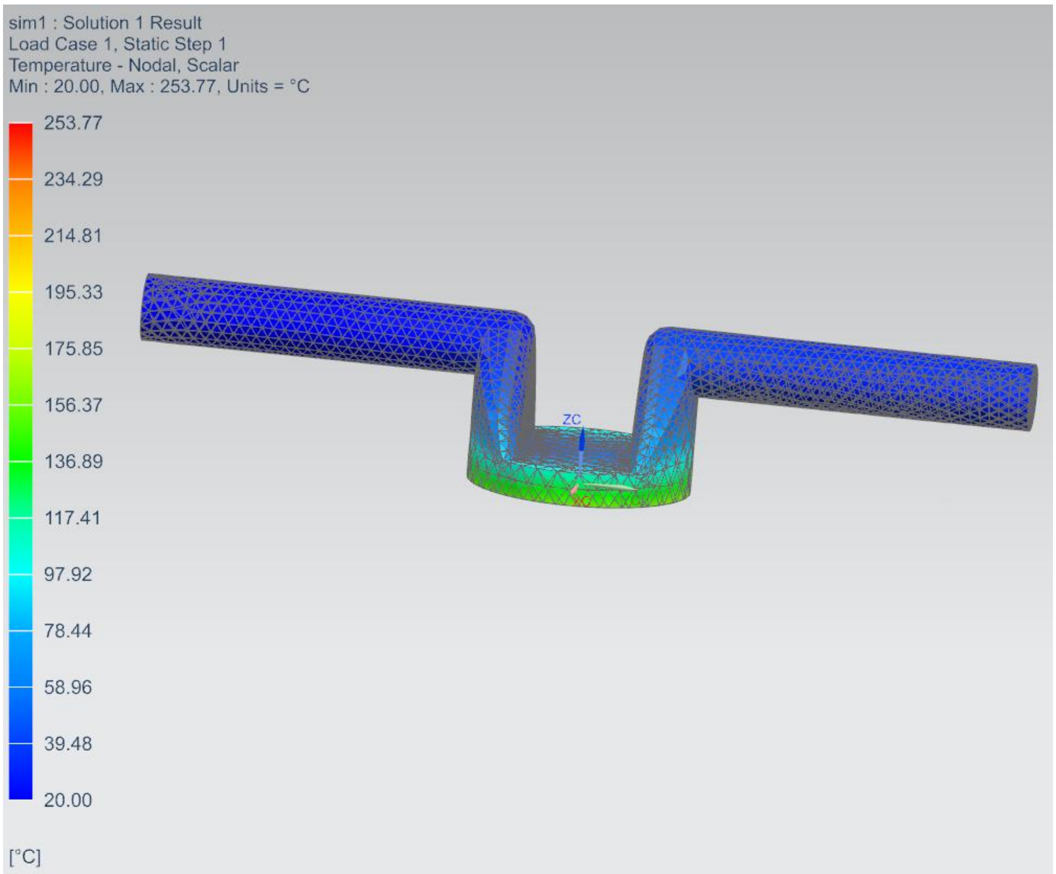
- Go to the *Post Processing Navigator*  . In the tree, expand (+) the *Solution 1* item, right-click on the *Thermal Flow* item, and select *Load* in the menu.
- Then, expand (+) the *Thermal Flow* item.

In what follows, we will visualize different results, such as the temperature of the pipe, the pressure, and the velocity field an temperature of the water.



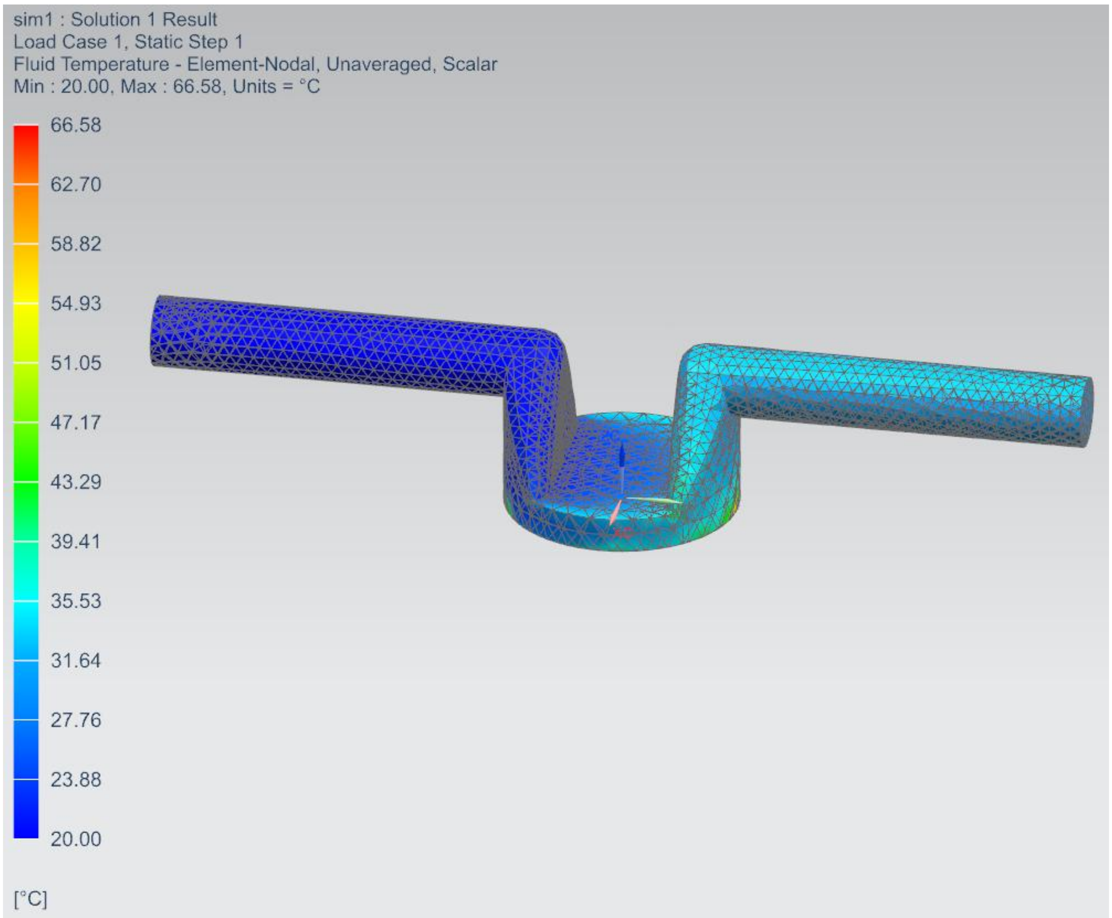
9 - Visualizing the temperature of the pipe.

- Right-click on the *Temperature-Nodal* item and select *Plot* in the menu. You should get the temperature of the pipe which varies between 20°C and 254°C. It can be observed that the maximal temperature is reached at the bottom circular face of the pipe, where the heat load is applied.



10 - Visualizing the temperature of the water.

- Right-click on the *Fluid-temperature...* item and select *Apply*. The water temperature varies between 20°C and about 67°C. It can be observed that the water temperature at the exit side of the pipe (right) is about 35°C.



11 - Visualizing the water pressure.

- In a similar way as done above, visualize the water pressure. There are three kinds of pressures: *total* pressure (P_{total}), *static* pressure (P_{static}) and *dynamic* pressure ($P_{dynamic}$).
- The *static* pressure (P_{static}) is the pressure applied by a fluid when it is at rest.

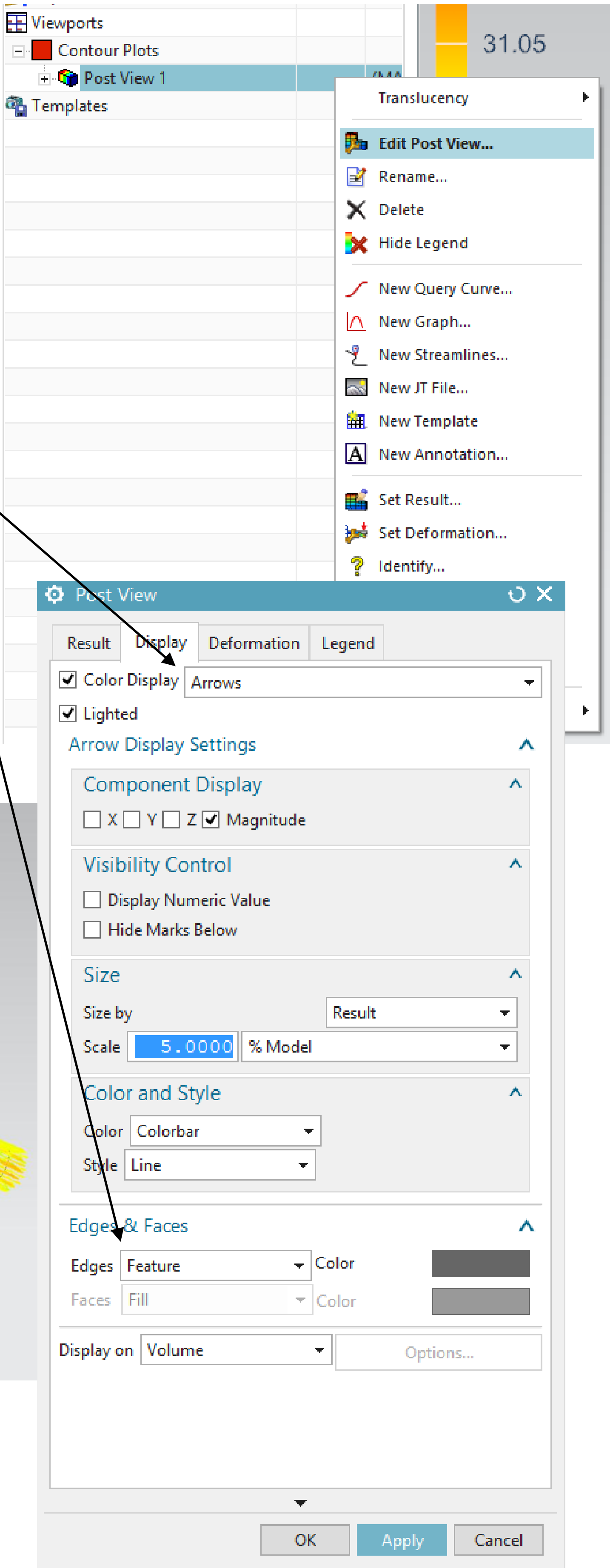
The *dynamic* pressure ($P_{dynamic}$) is the contribution to the total pressure by the movement of the fluid:

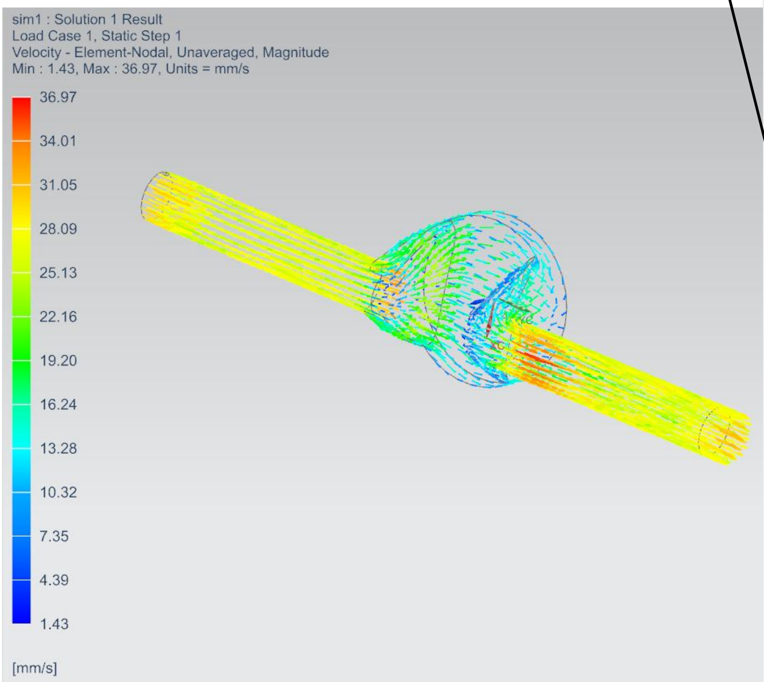
$P_{dynamic} = \rho \frac{\vec{v}^2}{g}$, where ρ is the fluid density, v its velocity and g = 9,81 m/s².

- The *total* pressure is given by: $P_{total} = P_{static} + P_{dynamic}$.

12 - Visualizing the velocity field.

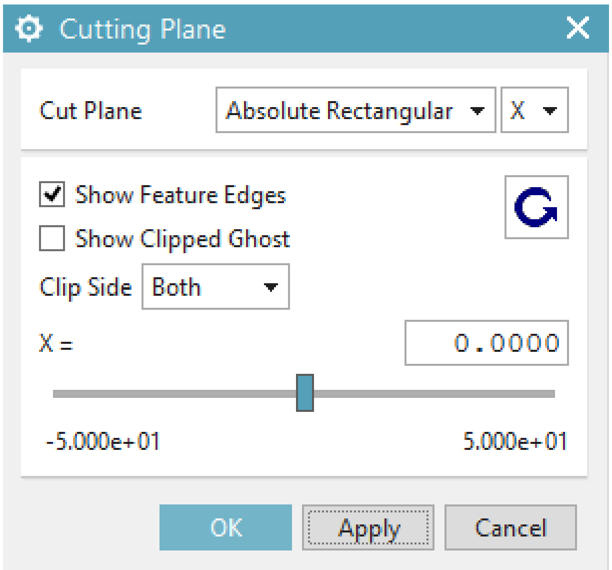
- Right-click on *Velocity...* and select *Apply*. By default, NX shows the magnitude of the velocity field, but it is also possible to visualize its directions and streamlines.
- In the *Post Processing Navigator*, select the *Post View...* item and right-click on it. In the menu, select *Edit Post View*.
- In the *Post View* dialog box, click on the *Display* tab and set the *Color Display* option to **Arrows**.
- In the *Edges & Faces* field, set the *Edges* option to **Feature** and click on *Apply* to validate (do not close the dialog box).
- You should get the result shown below.

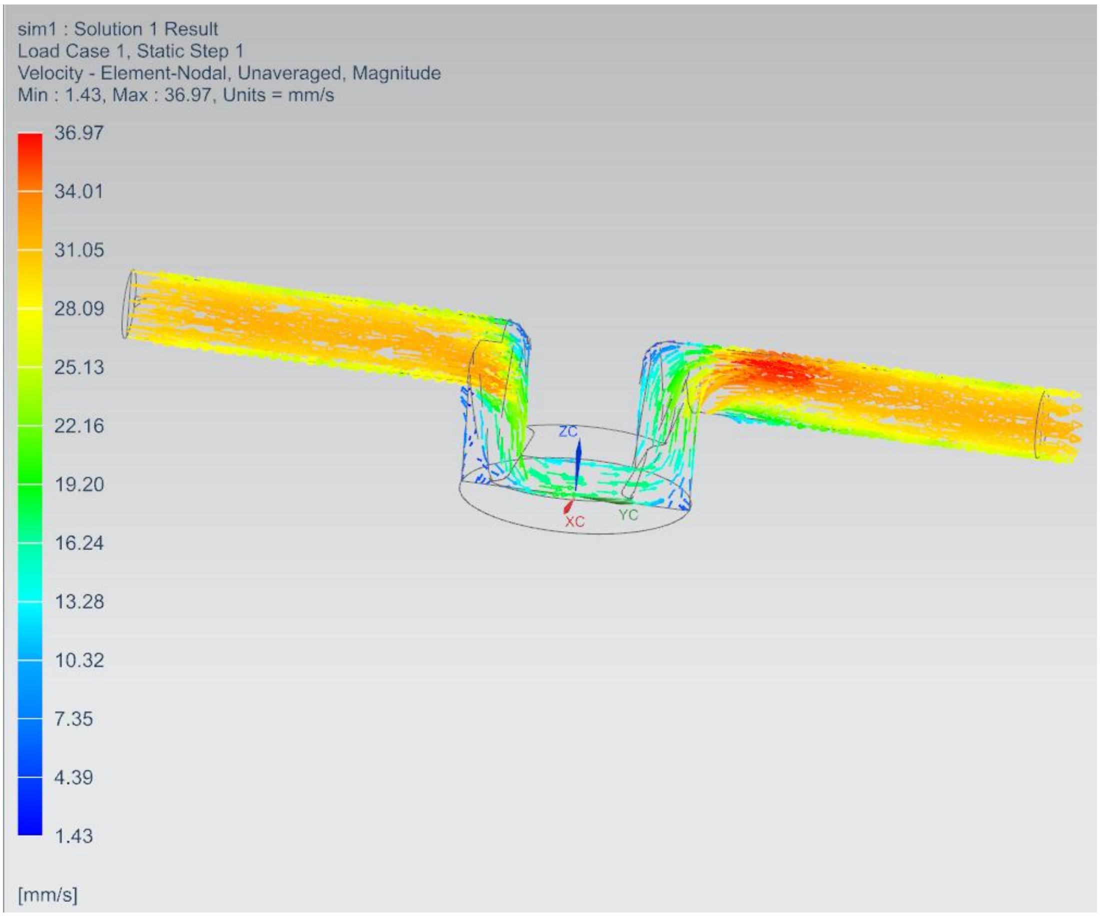




It is also possible to visualize the velocity field inside a given plane.

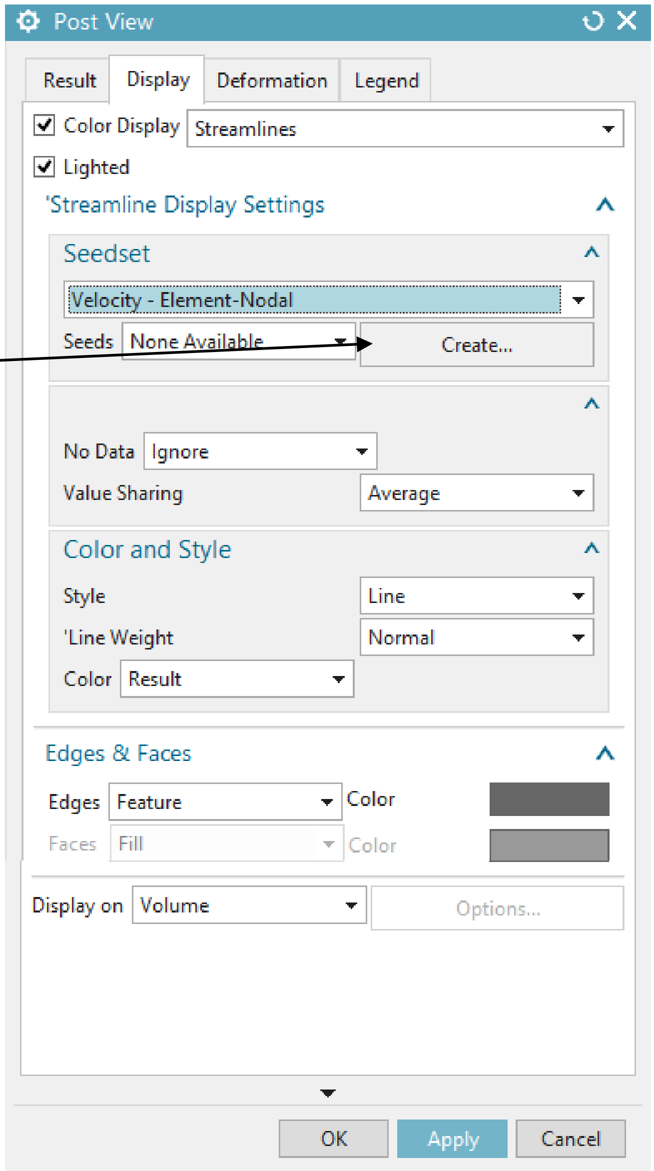
- At the bottom of the same dialog box, set the *Display On* option to **Cutting Plane**, and click on the *Options...* button.
- In the *Cutting Plane* dialog box, set the *Cut Plane* option to **Absolute Rectangular X**.
- Check the *Show Feature Edges* option and set the *Clip Side* option to **Both**.
- Finally, click *OK*, followed by *Apply* on the *Post View Dialog* box.
- You should obtain the velocity field at the middle of the pipe.

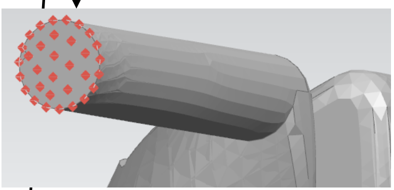


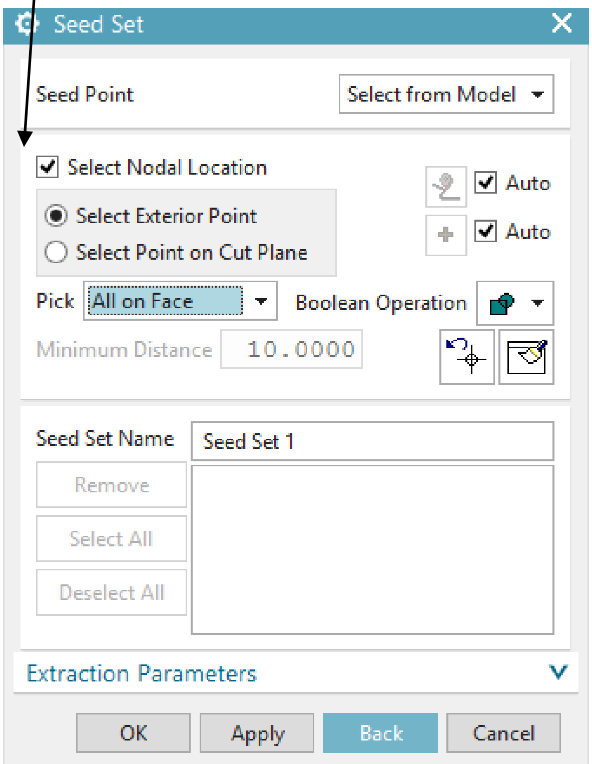


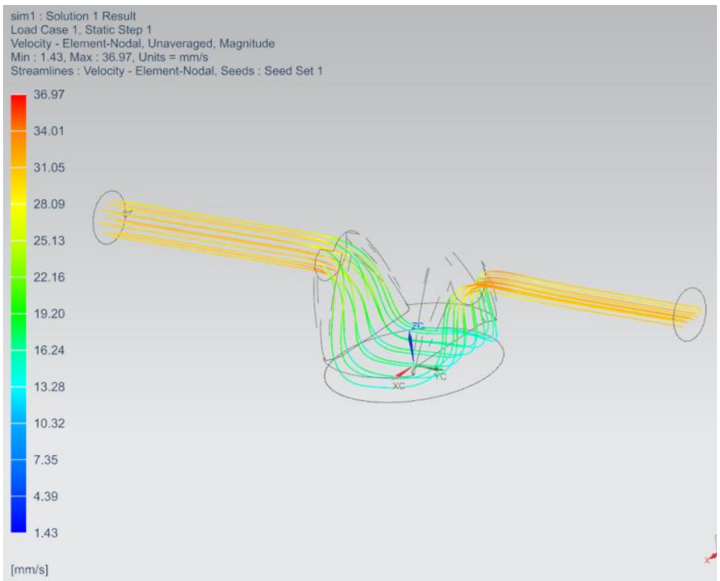
We will now visualize the streamlines.

- In the *Post View* dialog box, set back the option *Display On* to **Volume**.
- Set the *Color Display* option to **Streamlines**.
- In the *Seedset* field, click on the *Create...* button.
- In the *Seed Set* dialog box, check the *Select Nodal Location* check box, set the *Pick* option to **All on Face**, and click on *select Exterior Point*.
- Select the left (inlet) face of the pipe.
- Set the *Extract* option of the *Extraction Parameters* field to **Downstream** and click *OK*.
- In the *Post View* dialog box, click *OK*.
- You should get the below result.









We will now animate the streamlines with particles.

- Click on the *Results* tab of the toolbar, and then on the *Animate* 

button.
- In the *animation* dialog box, set the *Animate* option to **Streamline** and click on the *Play*  button.
- You should get particles travelling on the streamlines from the inlet (left) to the opening (right).
- **Do not forget to save your files.**