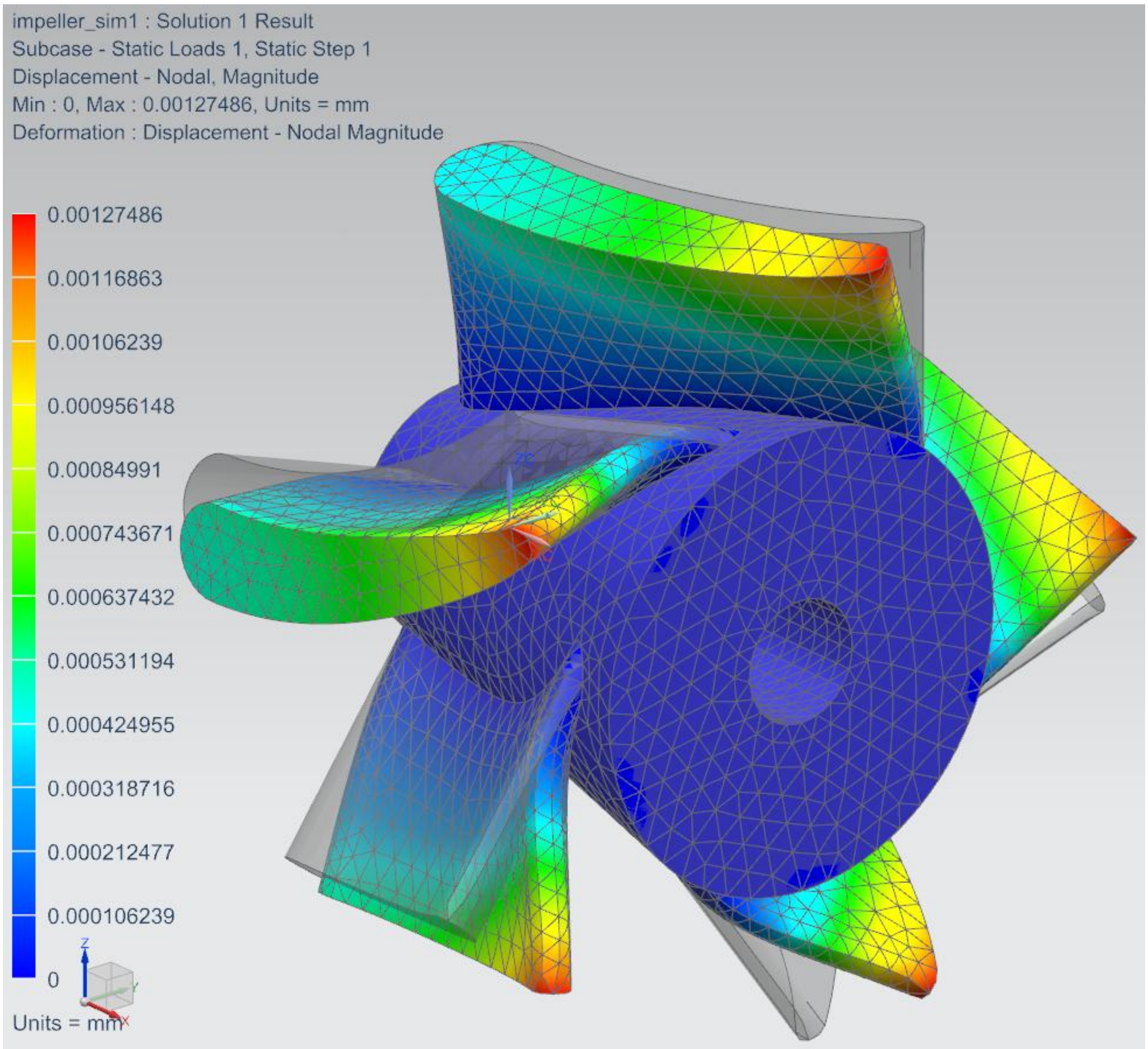
Using Siemens NX 11 Software

Finite Element Analysis - Impeller

Based on the tutorial "NX 10 for Engineering Design"[1].

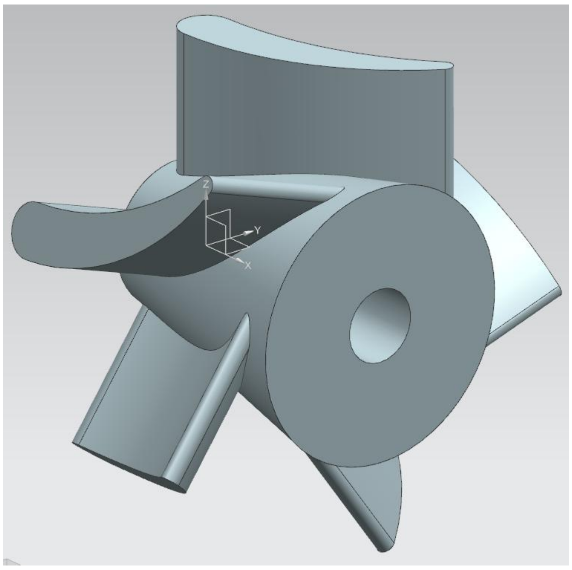


[1]*Ming C. Leu*
Amir Ghazanfari
Krishna Kolan
Department of Mechanical And Aerospace Engineering
University of Sience & Technology
Missouri

1 – Introduction.

The goal of this tutorial is to perform a finite element analysis (FEA) of the deformations of the blades of an impeller.

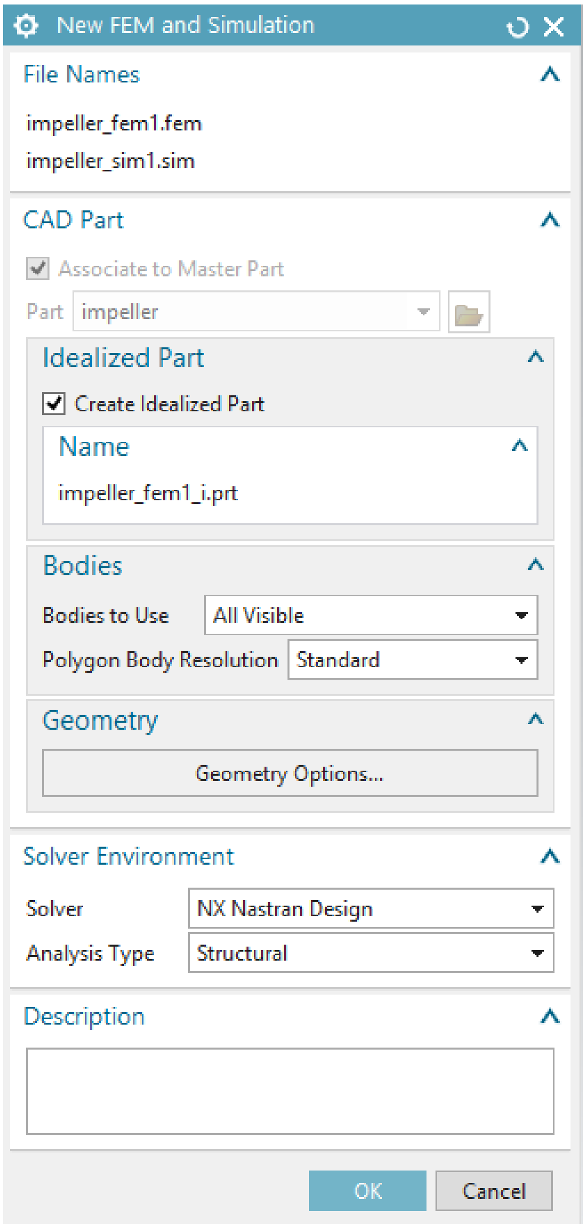
- Make a **copy** of the file *C:\Commun\NX \fem\impeller.prt* file in your local folder, and open it.



2 – Creating a new simulation.

- Click on the *Application* tab located above the toolbar, and then on the *Design* button 

.
- The *New FEM and Simulation* dialog box opens. In this dialog box, select in the *Solver Environment* field *NX Nastran Design* as *Solver* and *Structural* as *Analysis Type*.
- Click *OK* to validate.
- The *Solution* dialog box pops-up. Make sure to use de default options by clicking on the “Reset” button .
- Then, click *OK*.



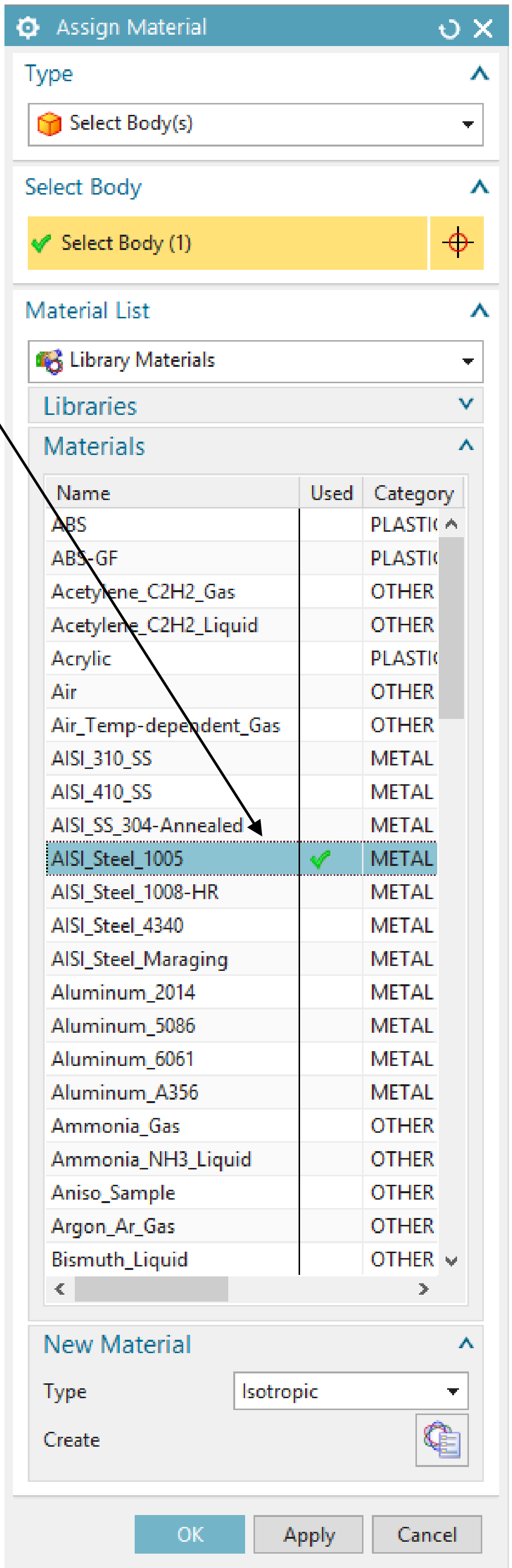
In what follows, we will perform the next couples of steps

1. Adding material properties for defining how the impeller will behave under some constraints.
2. Meshing the impeller from its CAD geometry.
3. Applying loads and boundary conditions.
4. Computing the FEA.
5. Visualizing the results.

3 – Material properties.

- Click on the *Assign Materials* button 

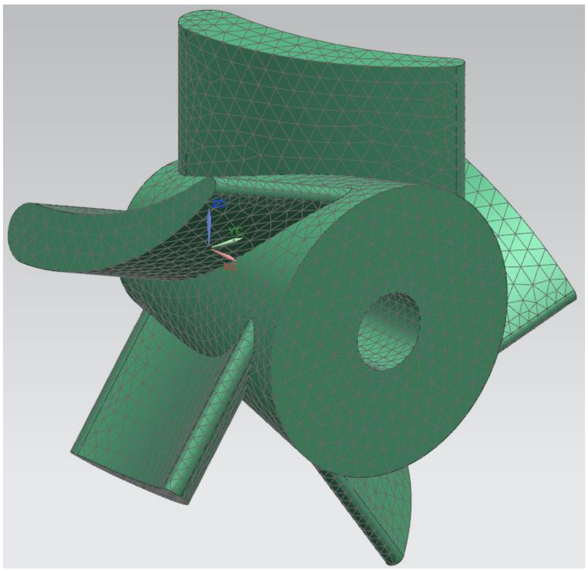
, and choose the *AISI_Steel_1005* material for the part.

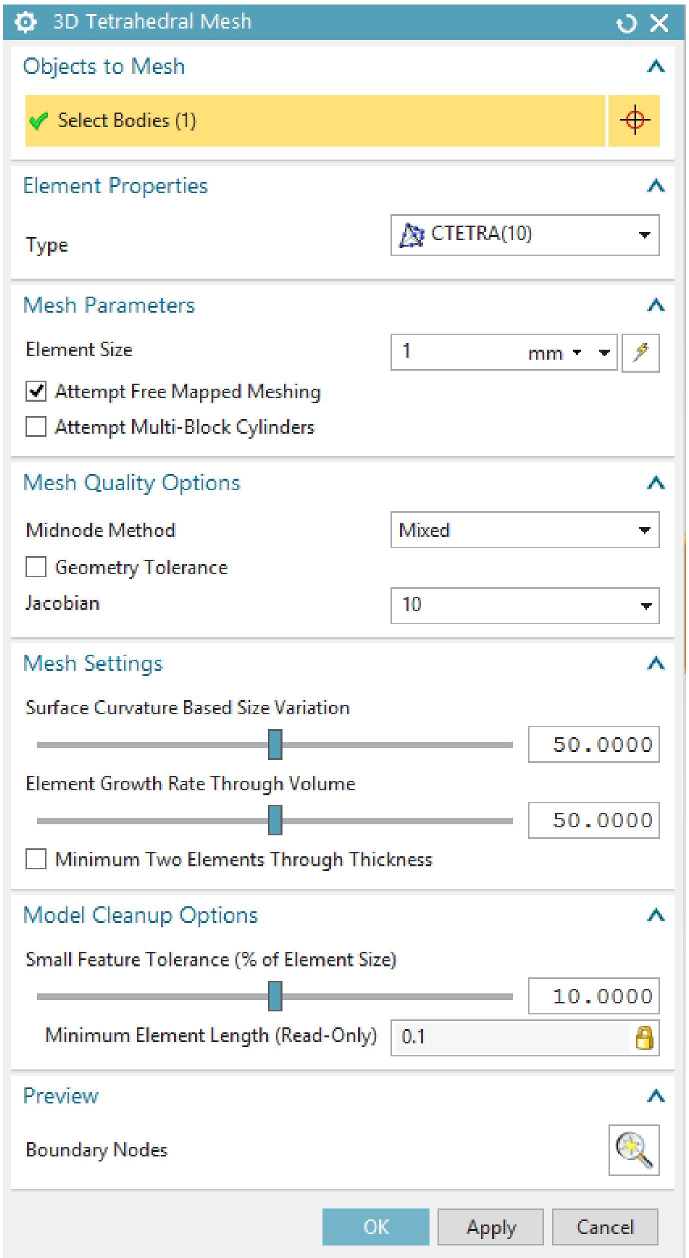


3 – Meshing.

- Click on the *3D Tetrahedral* button

.
- Select the part as *Object to Mesh* and use *CTETRA10* elements (tetrahedral elements with quadratic shape functions – 10 nodes).
- Set the *Element Size* to **1 mm** and click *OK* to validate.
- The meshing of the impeller will take a couple of seconds.





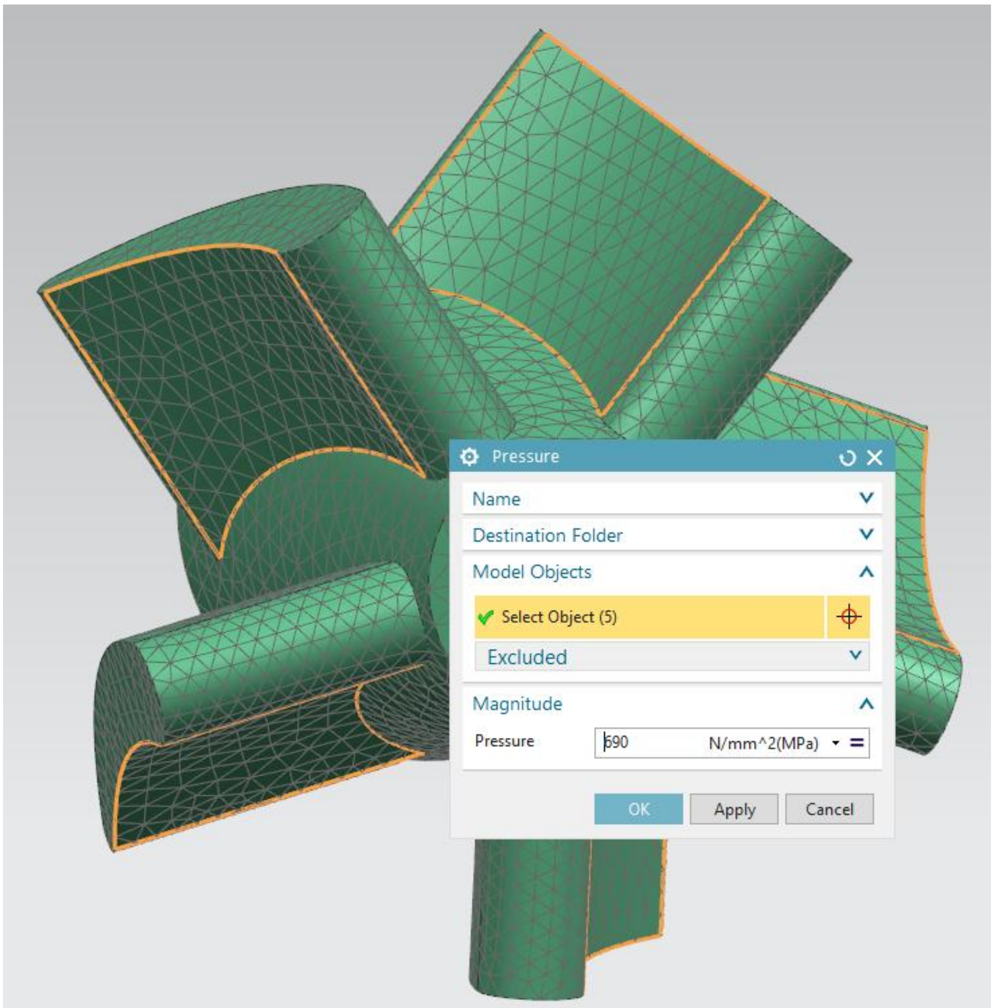
4 – Loads.

For the impeller, the major force acts on the concave surfaces of the turbine blades. This loading can be approximated by normal pressure on all the five surfaces.

- Click on the *Pressure* button 

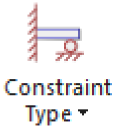
, under the *Load Type* button 

.
- In the *Pressure* dialog box, select all the five concave surfaces of the turbine blades and set a pressure of **0.690 MPa**.

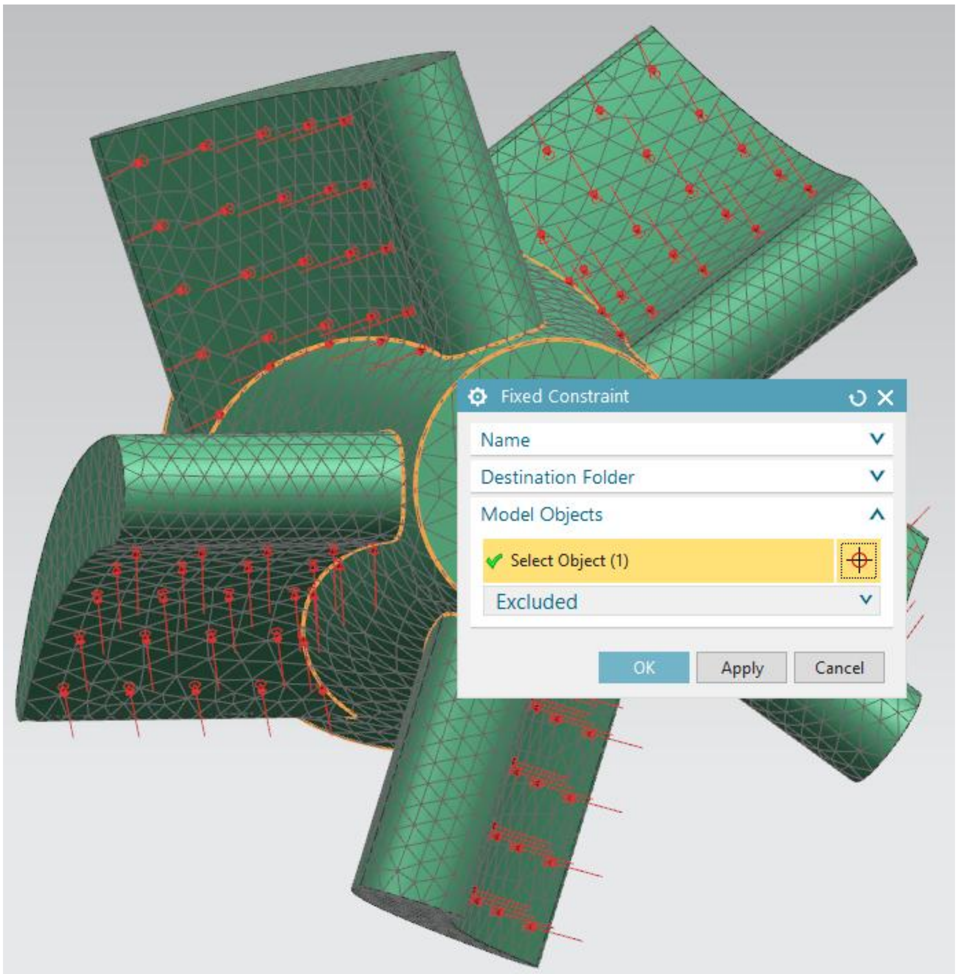


5 – Boundary conditions.

- Click on the *Fixed Constraint* button 

, under the *Constraint Type* button 

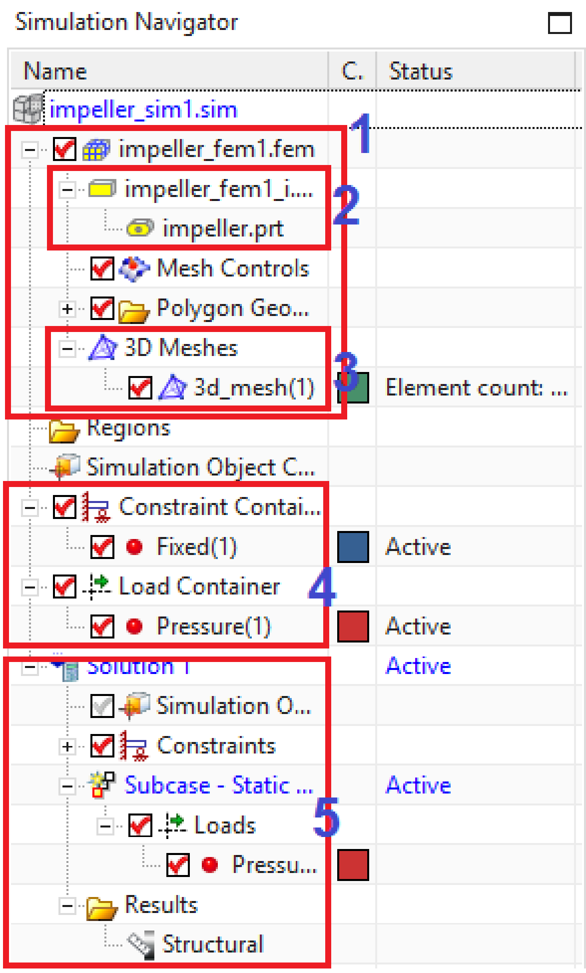
.
- Click on the conical surface of the impeller as shown in the figure below, and click *OK*.



6 – The simulation navigator.

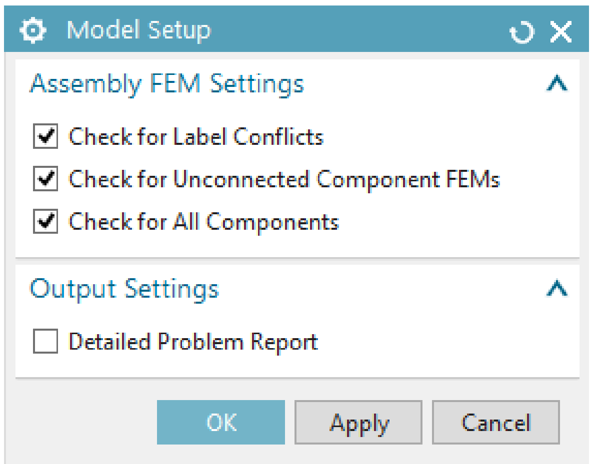
The *Simulation Navigator* regroups several items.

1. The FEM file which contains (a link to) the CAD geometry of the impeller and its corresponding mesh.
2. The CAD geometry of the impeller.
3. The associated mesh.
4. The loads and constraints that define the boundary conditions.
5. The solution and results of the simulation.



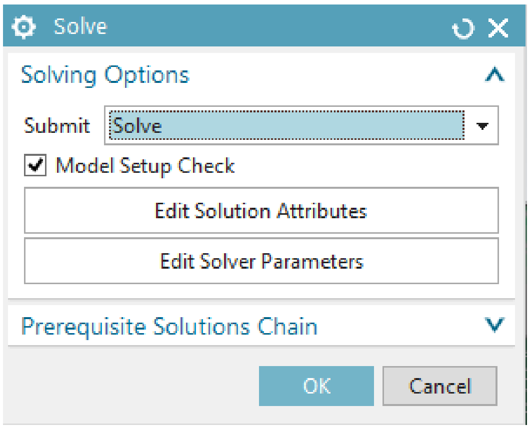
7 – Checking the consistency of the model.

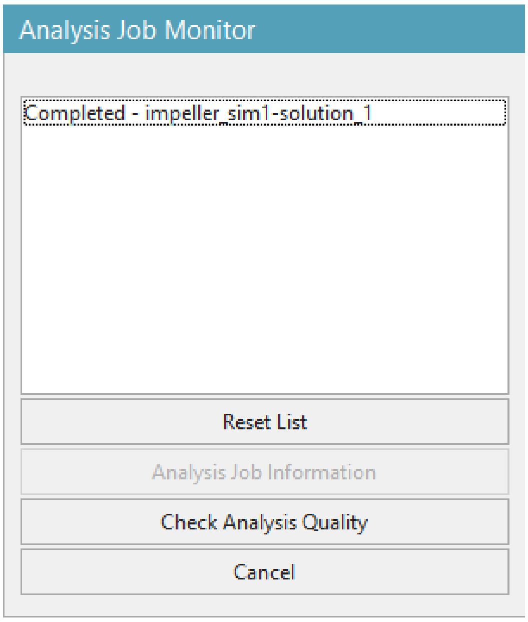
- Click on *Menu→Analysis→Finite Element Mode Check→Model Setup* Model Setup... .
- *In the Model Setup* dialog box, just click *OK*.
- This will display the result of the *Check*. You will be able to see any errors and warnings in a separate window. In case you get errors or warnings go back to the previous steps and complete the required things. If you do not get errors or warnings you are ready to solve the FEA problem.



8 – Solving.

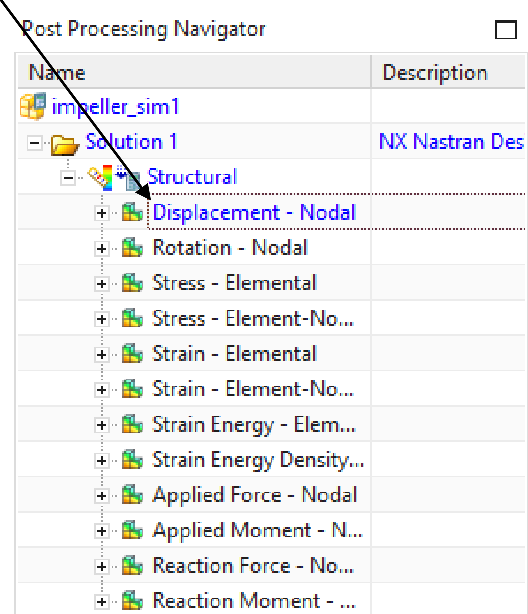
- Click on the *Solve* button .
- In the *Solve* dialog box, just click *OK*.
- This will generate the required files for the *NX Nastran* solver, which will compute the FEA and return the results to *Siemens NX*. (It may take a couple of seconds).
- Click on *Cancel* when the *Analysis Job Monitor* window shows *Completed*.
- Also, close the *Information* window.

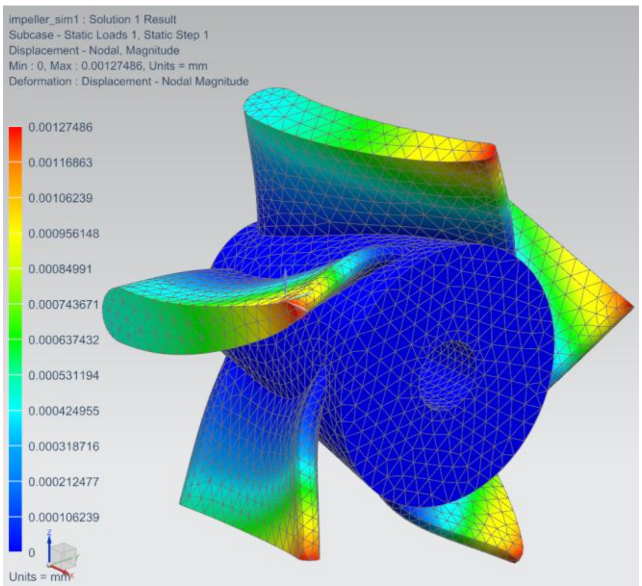




9 – Visualizing the results.

- In the *Simulation Navigator*, right-click on *Result* and then *Open*.
- This will open the *Post Processing Navigator*.
- You can, for instance, expand the *Structural* tree and double click on the *Displacement* item to see how the blades have been affected by the applied pressure.
- Maximum deformations appear in red at the tip of the blades and are about a magnitude of 0.001 mm.





10 – Animation.

- Click on the *Animate* button 

and set the number of frames to **100**, and the *Synchronized frame delay* to **2**.
- Click *OK* to launch the animation and click the *Stop* button for stopping it.
- For comparing the deformed situation from the undeformed one, click on the *Edit Post View* button 

.
- In the *Display* tab of the *Post View* dialog box, check the *Show undeformed model* and click *OK* to validate.
- You should obtain the result shown at the beginning of this tutorial.