CHAPTER 8 – FINITE ELEMENT ANALYSIS

Finite Element Analysis (FEA) is a practical application of the Finite Element Method (FEM) for predicting the response behavior of structures or fluids to applied factors such as forces, pressures, heats, and vibrations. Usually, the process starts with the creation of a geometric model. Then the model is subdivided (meshed) into small pieces (elements) of simple geometric shapes connected at specific node points. The material properties and the boundary conditions are applied to each element. Finally, software such as NX 12 solves this FEA problem and outputs results and visualizations. It helps engineers to have a better understanding of the product performance before it is fabricated and tested.

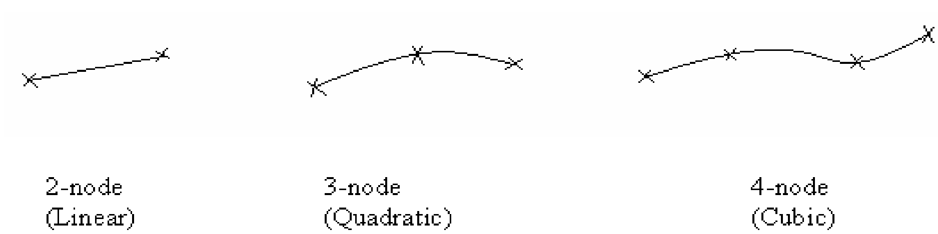
Some of the applications of *FEA* are *Structural Analysis, Thermal Analysis, Fluid Flow Dynamics*, and *Electromagnetic Compatibility*. Of these, *FEA* is most commonly used in structural and solid mechanics applications for calculating the mechanical behavior (e.g. stresses and displacements). These are often critical to the performance of the hardware and can be used to predict failures. In this chapter, we are going to deal with the structural stress and strain analysis of a solid part.

8.1 OVERVIEW

8.1.1 Element Shapes and Nodes

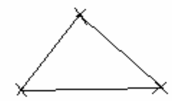
The elements can be classified into different types based on the number of dimensions and the number of nodes in an element. The following are some of the types of elements used for discretization.

One-dimensional elements



Two-dimensional elements

Triangular:

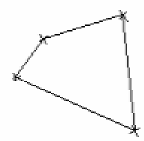


3-node
(Linear)

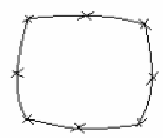
6-node
(Quadratic)

10-node
(Cubic)

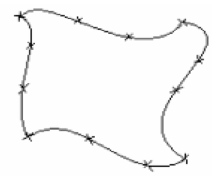
Quadrilateral:



4-node
(Linear)



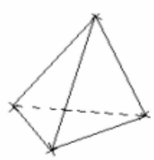
8-node
(Quadratic)



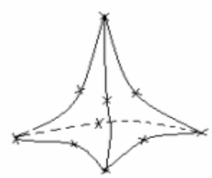
12-node
(Cubic)

Three-dimensional elements

Tetrahedral (a solid with 4 triangular faces):



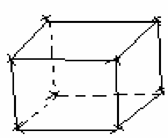
4-node
(Linear)



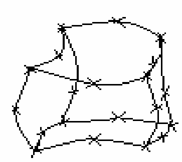
10-node
(Quadratic)

20-node
(Cubic)

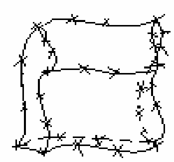
Hexahedral (a solid with 6 quadrilateral faces):



8-node
(Linear)



20-node
(Quadratic)



32-node
(Cubic)

Types of nodes

- Exterior nodes
  - Corner nodes
  - Side nodes
- Interior nodes

Usually, FEA can have a more accurate solution as the size of finite element becomes smaller, but the computing time becomes longer as well.

8.1.2 Solution Steps

Starting the Simulation: You can select a solver from one of these: NX Nastran, NX Nastran Acoustic, NX Nastran Vibro-Acoustic, NX Nastran Design, Samcef, NX Thermal/Flow, Simcenter Electronic Systems Cooling, Simcenter Space Systems Thermal, NX Multiphysics, Simcenter Acoustics BEM, MSC Nastran, Ansys, Abaqus, and LS-DYNA. In addition, you can choose the type of analysis to be performed. In this tutorial, only Structural Analysis will be covered with NX Nastran Design.

Choosing the Material Properties: This allows you to change the material properties that will be assigned to the model. For example, if we use steel to manufacture the impeller, we can enter the material properties such as density, Poisson's ratio, etc. These material properties can also be saved in the library for future use or can be retrieved from Library of Materials.

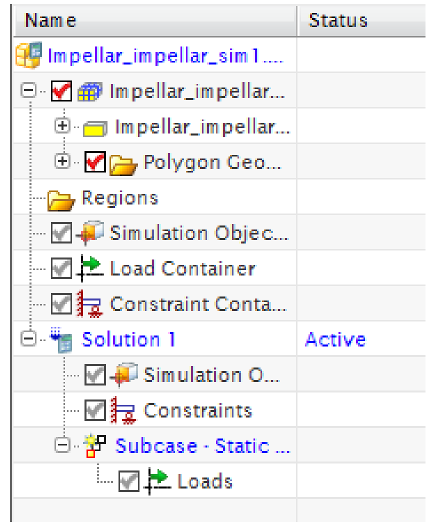
Applying the Loads: This option allows you to apply different types of loads, such as forces or pressures on the solid along with the directions and magnitudes.

Applying the Boundary Conditions: Simply speaking, boundary conditions constrain the degrees of freedom of the elements. Some elements can be rotationally fixed and some can be constrained from translational movement.

Meshing the Bodies: This is used to discretize the model into finite elements. Normally, we select tetrahedral shapes of elements for approximation. You can still select the 2-D and 1-D elements depending on the situation and requirements by choosing these options from the drop-down menu.

Solution and Results: This is the command to solve all the governing equations by the selected solver and all the above options. This solves and gives the results of the analysis of the problem.

8.1.3 Simulation Navigator



The *Simulation Navigator* provides the capability to activate existing solutions, create new ones, and use the created solution to build mechanisms by creating and modifying motion objects. To display the *Simulation Navigator*,

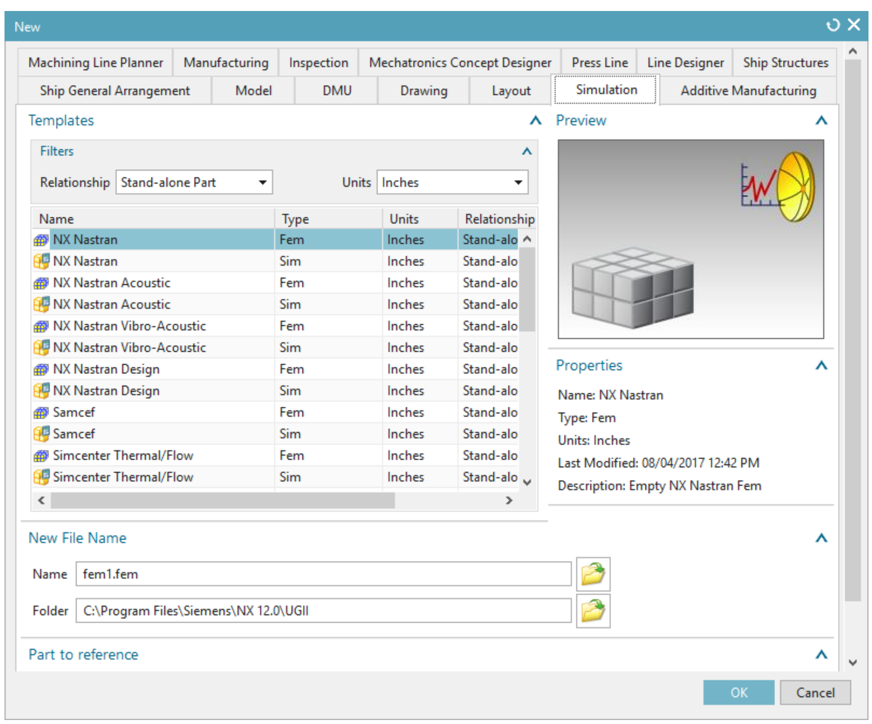
- ➢ Click the **Simulation Navigator** tab in the **Resource bar** as shown in the figure

It shows a list of the simulations created for the model. In each simulation, it displays a list of loads, boundary conditions, types of meshes, results, reports generated and so on.

8.2 SIMULATION CREATION

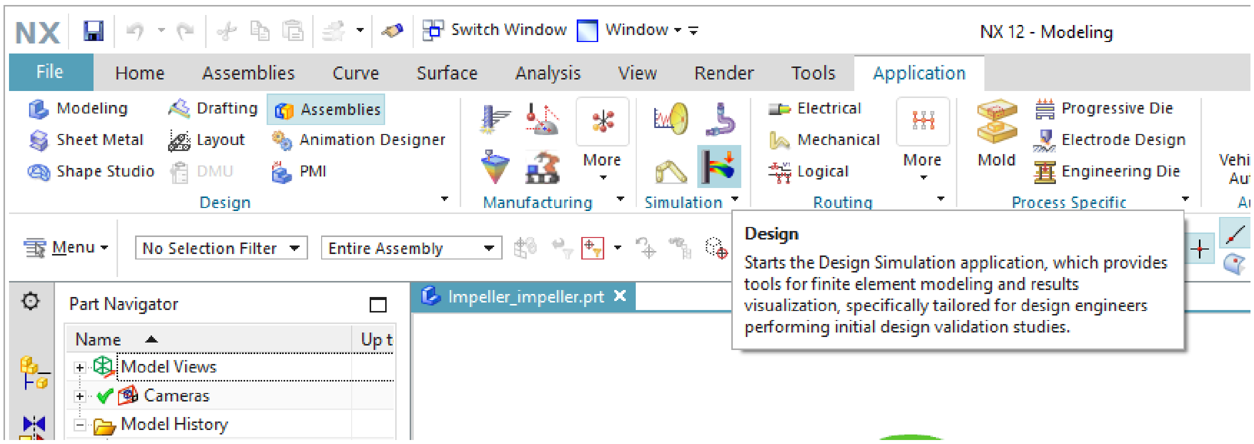
- ➢ Copy and paste the file **Impeller_impeller.prt** into a new folder to avoid changes being made to the assembly

- ➢ Click on **New** → **Simulations** if the part is NOT already opened in the NX window



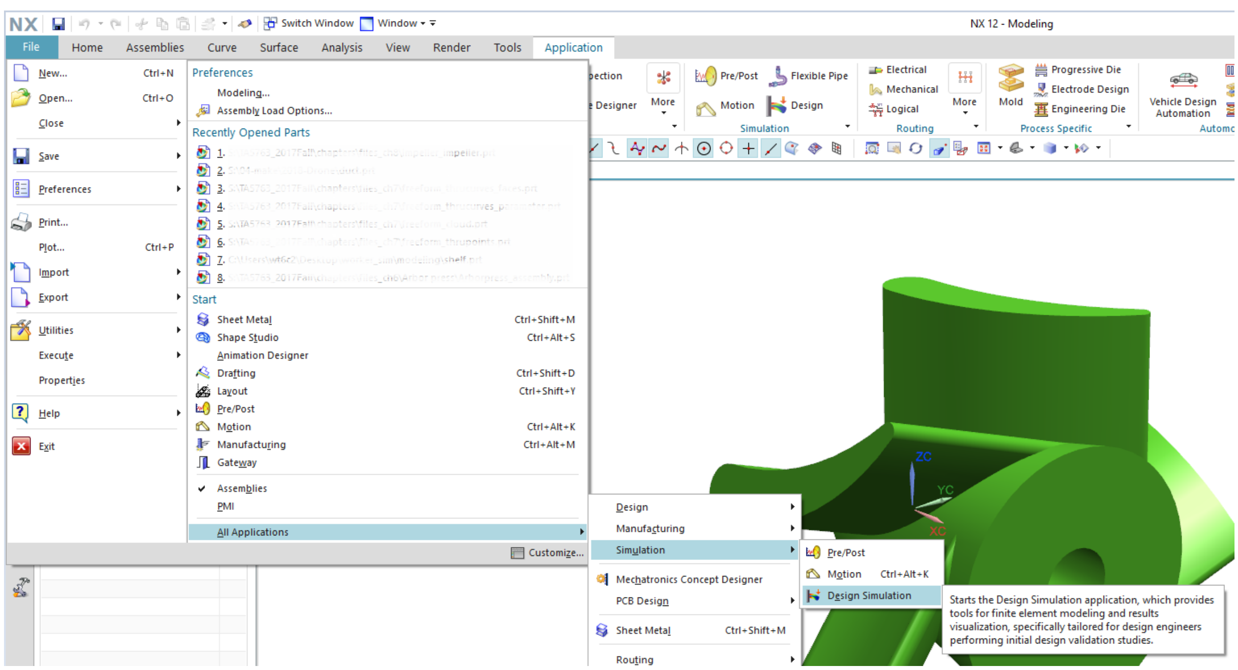
- ➢ Open this newly copied file

➢ If the part is already opened in NX, then from the top ribbon bar, click on **Application** → **Design**



Or

➢ click on **File** → **All Applications** → **Simulation** → **Design Simulation**



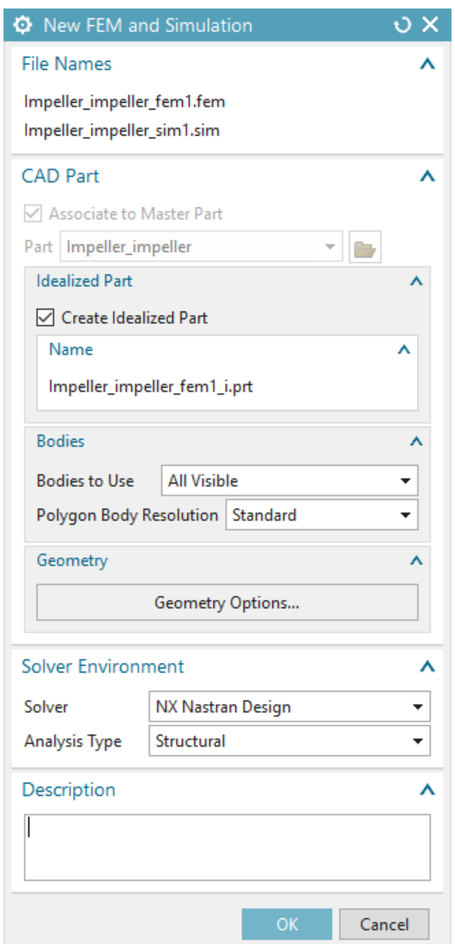
When you first open any file in *Design Simulation* module, it will automatically pop up the New FEM and *Simulation* dialog to create a simulation.

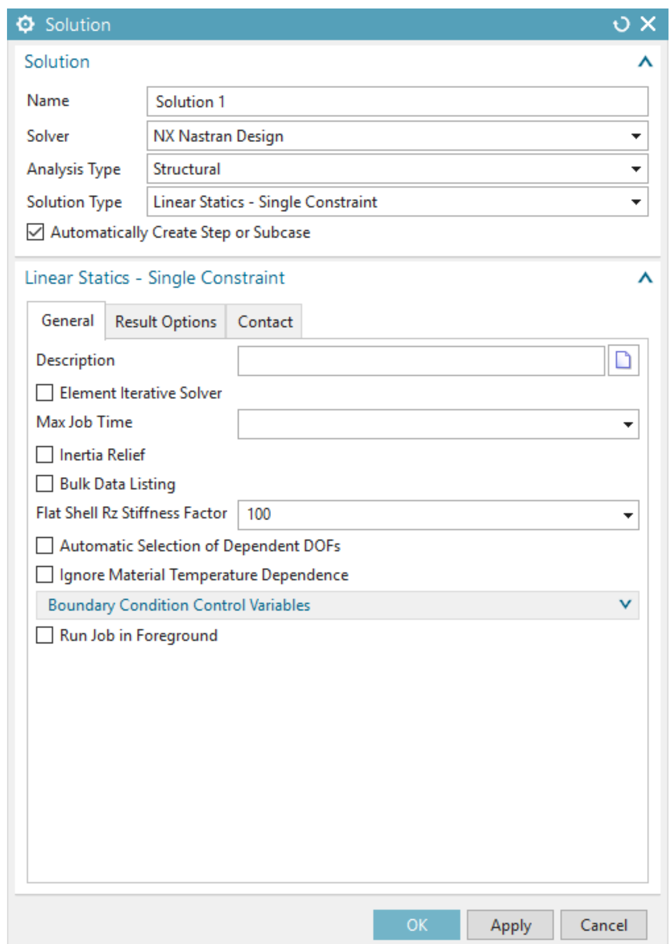
- In the popup dialog, click OK to create a new simulation.

Then in the next popup *Solution* window, you can select the *Solver* and the *Analysis Type*.

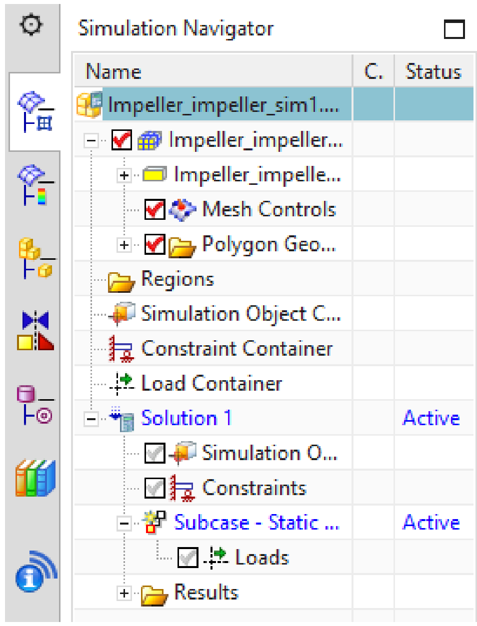
The default Solver type is *NX Nastran Design* and Analysis type is *Structural*.

- Choose **OK** to create a new **Solution** called Solution 1, which will be displayed in the **Simulation Navigator.** Here we keep the other items as default.





Now the Simulation Navigator will look like the following figure.



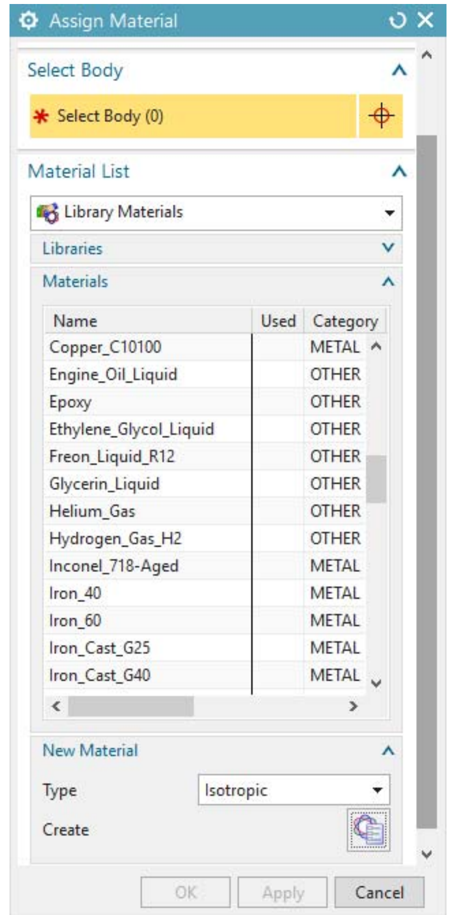
8.3 MATERIAL PROPERTIES

The next step is to assign the material properties to the solid model for this simulation. Because we do not have any data in the library to retrieve for standard material, we will create one. Let us assume that we will use steel to manufacture the impeller.

- ➢ Click on **Assign Materials** from the ribbon bar shown above

The *Assign Material* window will pop up. You have the option of choosing the pre-defined materials from the *Library* or create a new material.

- ➢ Select the **Impeller** for **Select Body**
- ➢ Click on the **Create** icon to create a new material

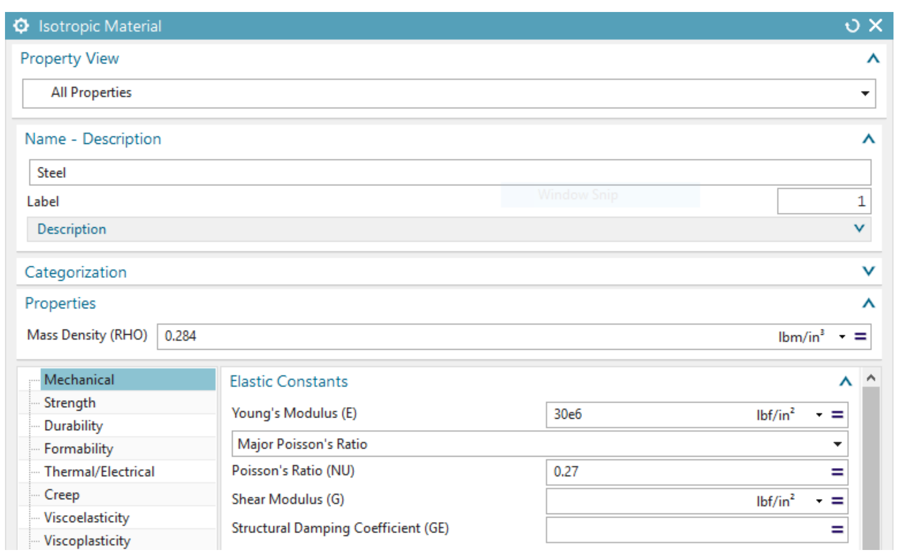


Enter the name and values as shown in the following figure. Pay attention to the units.

(Note that 30e6 represents 30×10^6)

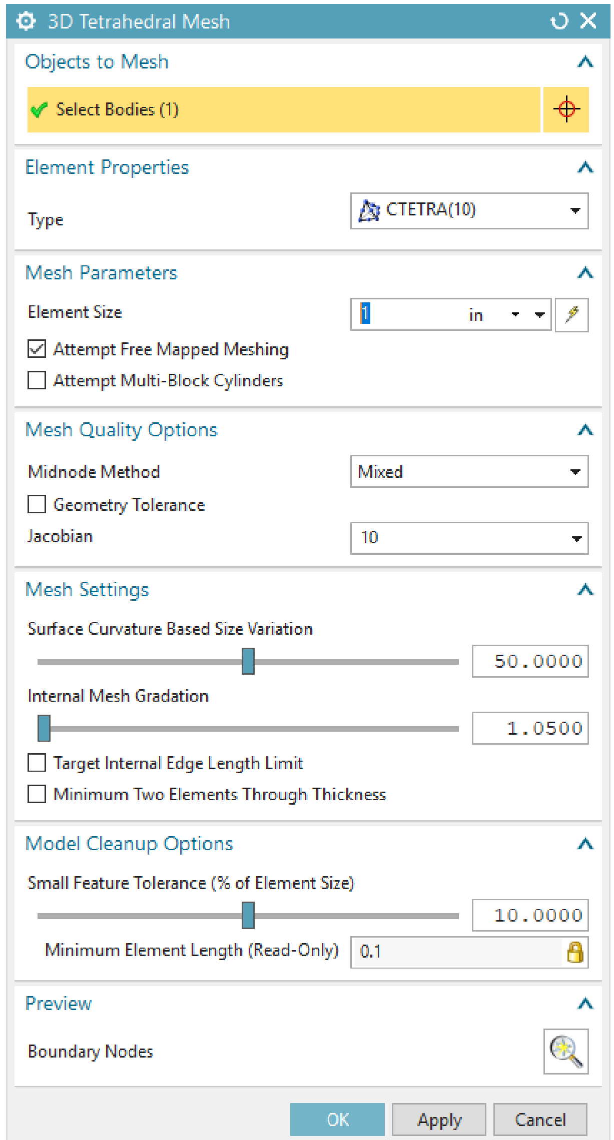
- ➢ Choose **OK** to exit the **Isotropic Material** window

Now we have assigned the material to the impeller model.



8.4 MESHING

The *Mesh* option discretizes the model into small elements.



- Click on the **3D Tetrahedral** icon 

A window will pop up asking for the type and size of the elements.

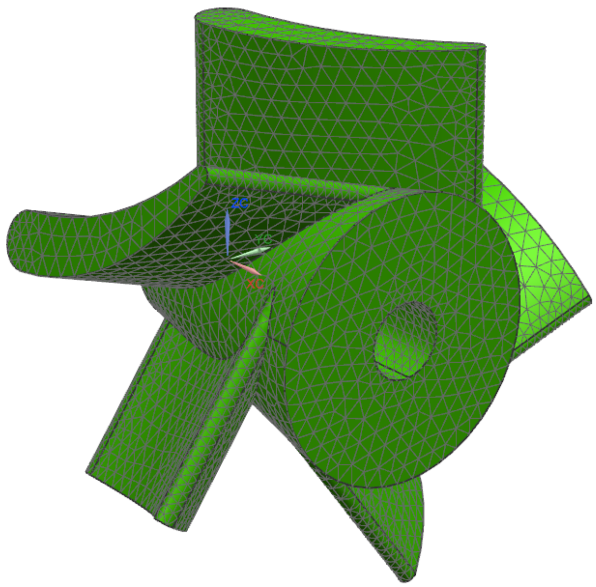
- First, click on the **impeller model** on the screen for **Select Bodies**.

Then, there are two types of *Tetrahedral Elements* available in NX 12. One is 4-node and the other is 10-node.

- Choose the **Type** to be **TETRA(10)**
- Enter the **Element Size** as **1.0 inch**
- Click **OK**

You can find the model with small tetrahedral elements. It will look like the figure shown below.

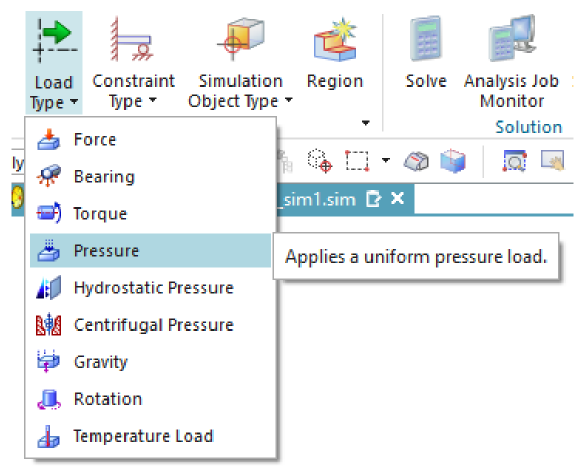
Note: While meshing the solid, there is a trade-off you need to consider. If you choose a smaller element with higher nodes you will get better accuracy in your analysis than larger element. However, the time required to solve the model with smaller elements will much greater than with larger element. Hence, based on the accuracy requirement of the study and how critical the component is in terms of the end product, choose the appropriate parameters for the elements and nodes.



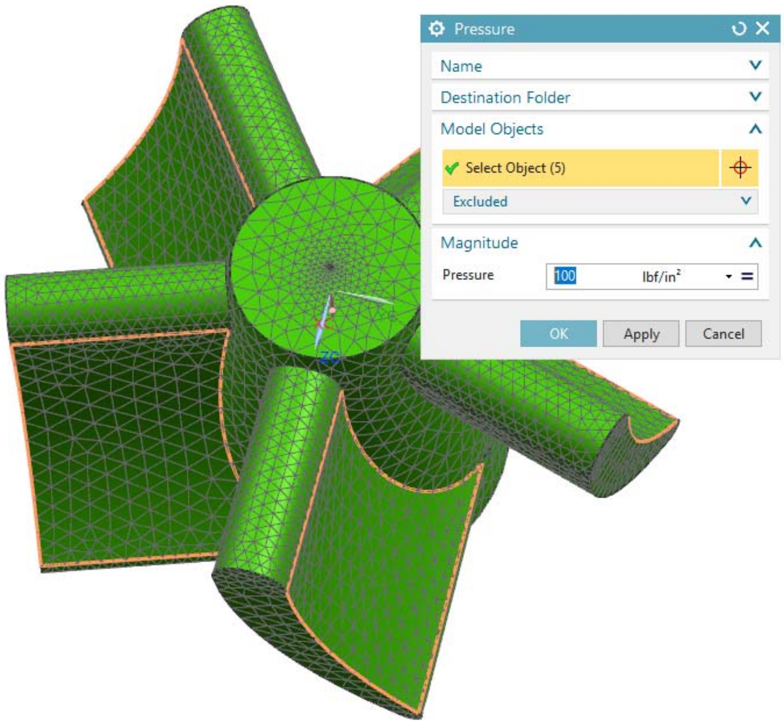
8.5 LOADS

The loads applied on the solid model should be input to the system. For the impeller, suppose 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. Since we are not concerned about the magnitude of the load, let us take the value to be 100 lbf/in^2 inch to exaggerate the deformation of the blades.

- Click on **Load Type** and choose **Pressure**



- Click on the five concave surfaces of the blades as shown in the following figure
- Enter the value for **Pressure** as **100** and keep the unit as lbf/in^2 (psi)

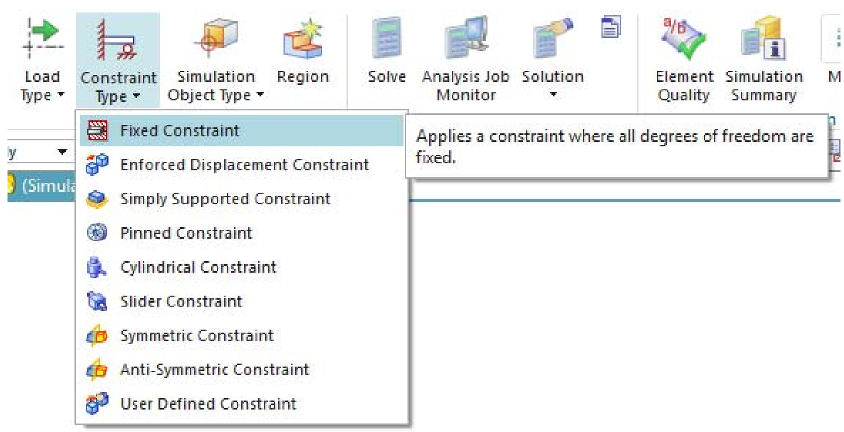


8.6 BOUNDARY CONDITIONS

The impeller rotates about the axis of the cone with the shaft as you can see in the assembly in the previous chapters. It is not fixed but our concern is the deformation of the blades with respect to the core of the impeller. The conical core is relatively fixed and the deformations of the blades are to be analyzed accordingly.

➢ Click on the **Constraint Type** icon

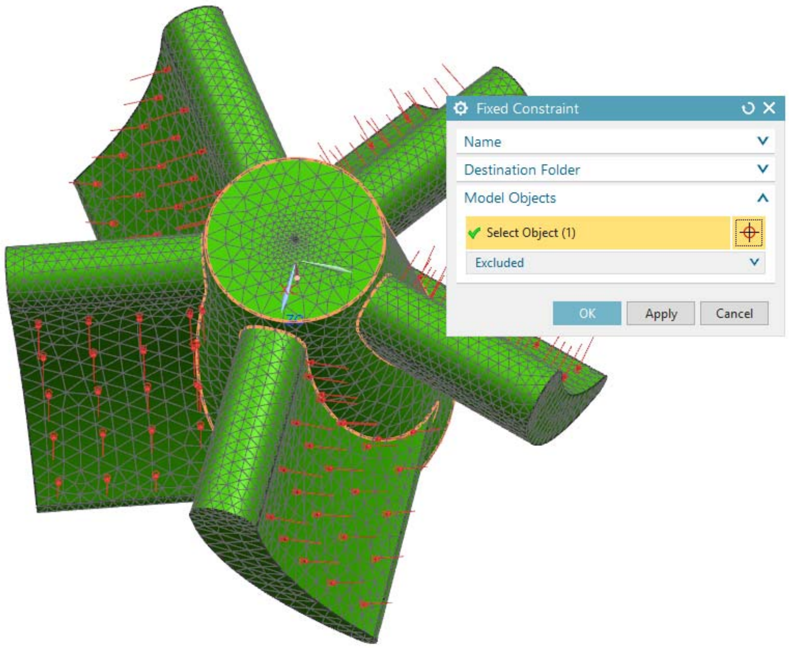
➢ Select the **Fixed Constraint**



This type of constraint will restrict the selected entity in six DOF from translating and rotating. You can see the different constraints available by clicking the *Constraint Type* drop-down menu on the toolbar.

➢ Click on the conical surface of the impeller as shown in the following figure

➢ Click **OK**



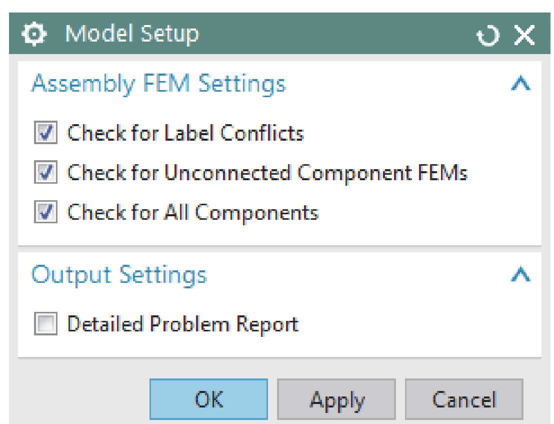
8.7 RESULT AND SIMULATION

8.7.1 Solving the Simulation

The Finite Element Model is now ready for solving and analysis. It is a good practice to first check for model completion before we get into solving the model. To check the model

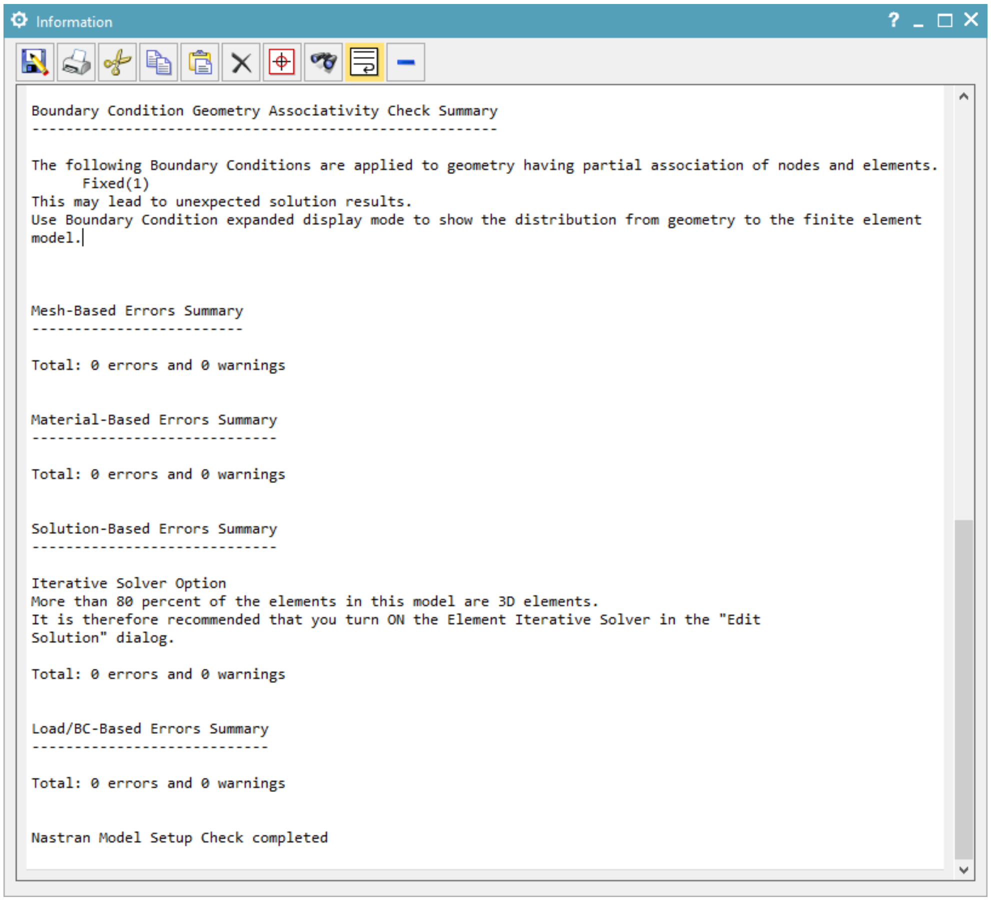
➢ Click on the **Menu** → **Analysis** → **Finite Element Mode Check** → **Model Setup** or click the **Model Setup** icon in the **Checks and Information** group in the ribbon bar

This will pop up a window as shown on the right.



➢ Choose **OK**

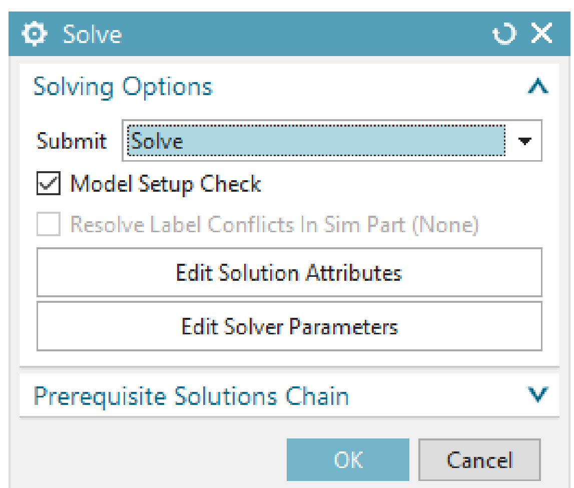
This will display the result of the *Check* operation**.** You will see any errors and warnings in a separate window. In case you get errors or warnings go back to the previous steps and complete the required things. If you do not get errors or warnings you are ready to solve the FEA problem.



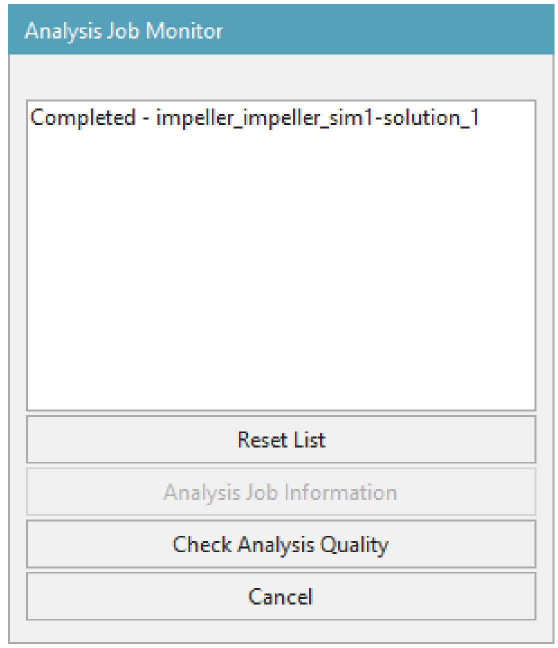
➢ Click on the **Solve** icon 

This will open the *Solve* window.

➢ Click **OK** without making any changes



It may take a while to solving the job. Wait until the *Analysis Job Monitor* window appears, showing the job to be *Completed*. While the solver is doing computations, the *Analysis Job Monitor* will show as *In Progress*

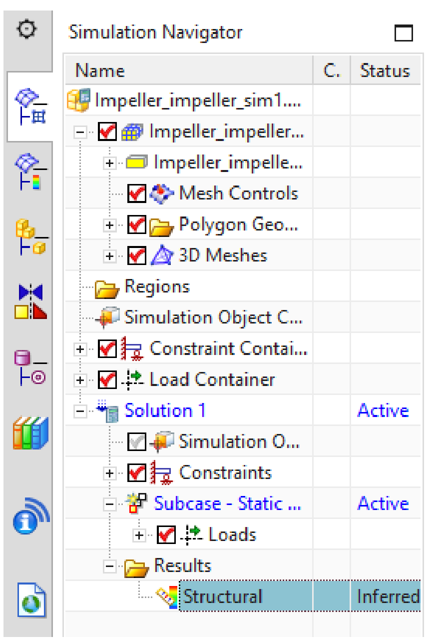


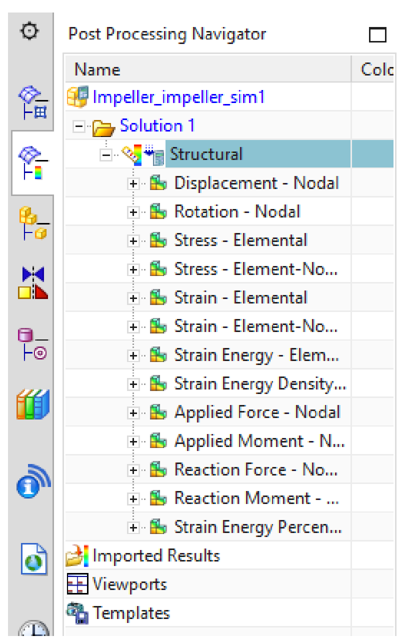
- Click on **Cancel** when the **Analysis Job Monitor** window shows **Completed**

8.7.2 FEA Result

- From **Simulation Navigator**, double click on **Structural** under **Results**
- You will be directed to the **Post Processing Navigator**

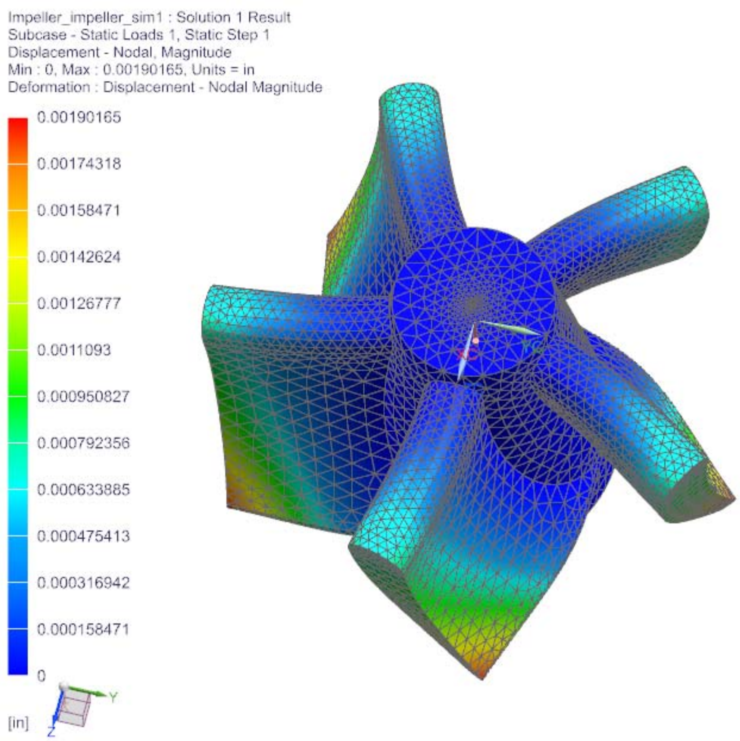
The *Post-Processing Navigator* shows all the *Solution* you just solved. If you click the '+' sign in front of the *Solution*, you will see the different analyses that have been performed on the model.



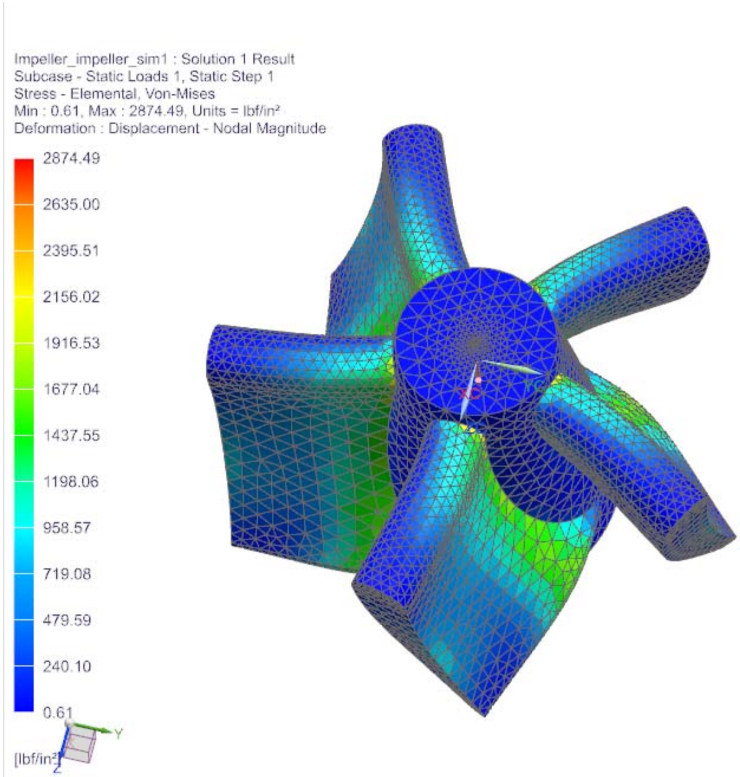


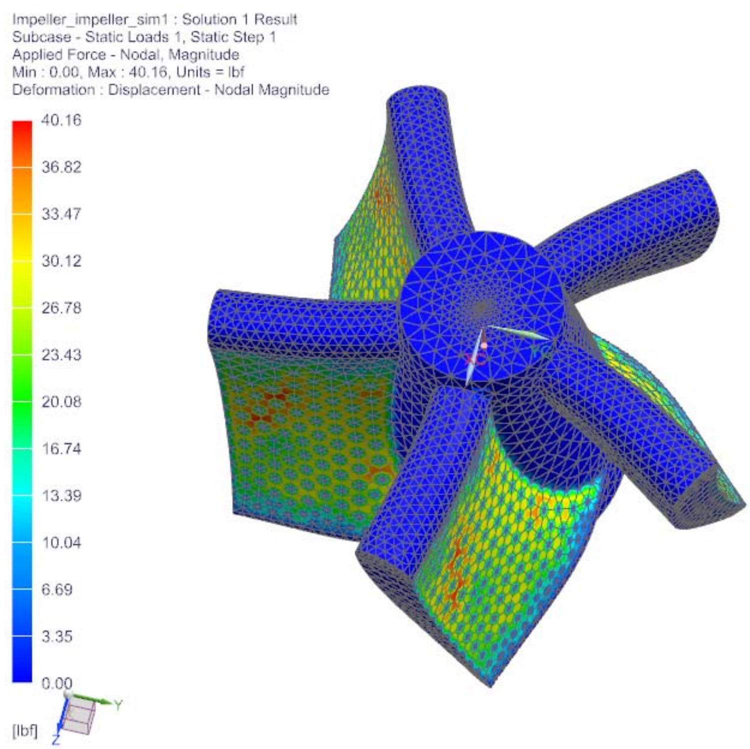
➢ Double-click on the **Displacement-Nodal**

The screen will now appear as shown below. You can easily interpret the results from the color-coding. The orange-red color shows the maximum deformation zones and the blue area shows the minimum deformation zones. You can observe that because the conical core is fixed, it experiences zero deformation. The analysis also shows that the maximum deformation experienced at the tip of the blades is approximately 1.9×10^{-3} inches.



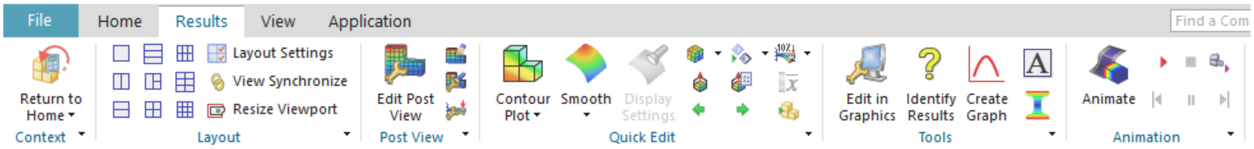
On the *Post-Processing Navigator*, you can keep changing the results by double clicking each option as shown below. You can click on the other inactive marks to see various results. Some of the other results are shown below.

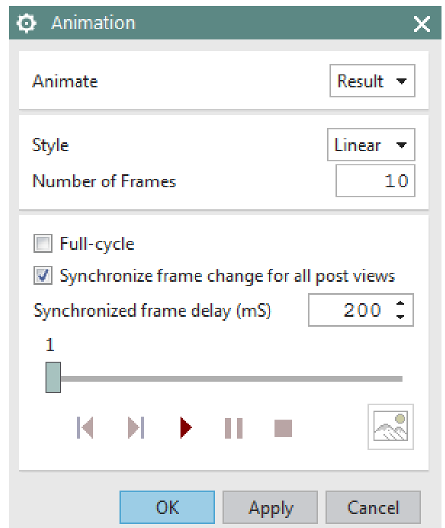

Impeller_impeller_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.61, Max : 2874.49, Units = lbf/in²
Deformation : Displacement - Nodal Magnitude
2874.49
2635.00
2395.51
2156.02
1916.53
1677.04
1437.55
1198.06
958.57
719.08
479.59
240.10
0.61
[lbf/in²]
Y
Z


Impeller_impeller_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Applied Force - Nodal, Magnitude
Min : 0.00, Max : 40.16, Units = lbf
Deformation : Displacement - Nodal Magnitude
40.16
36.82
33.47
30.12
26.78
23.43
20.08
16.74
13.39
10.04
6.69
3.35
0.00
[lbf]
Y
Z

8.7.3 Simulation and Animation

Click on the **Results** tab from the top ribbon bar. A group for **Animation** can be seen on it as follows



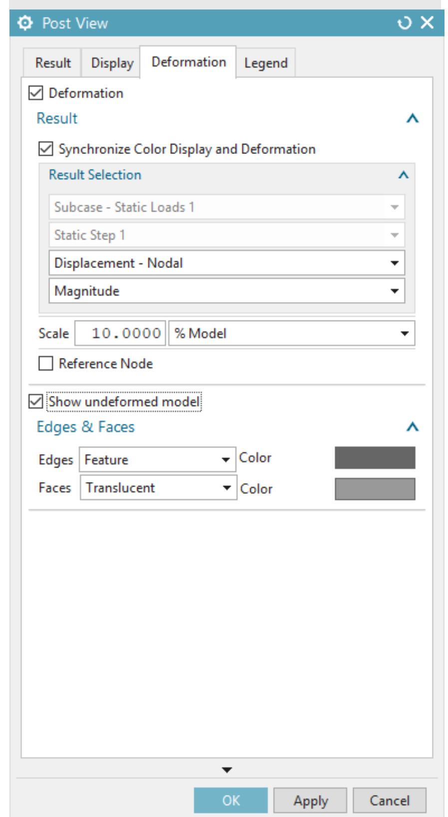


- Click on the **Animate** icon.
- In the **Animation** window, change the number of frames to **10** and click on the **Play** button  to see the animation of the deformation

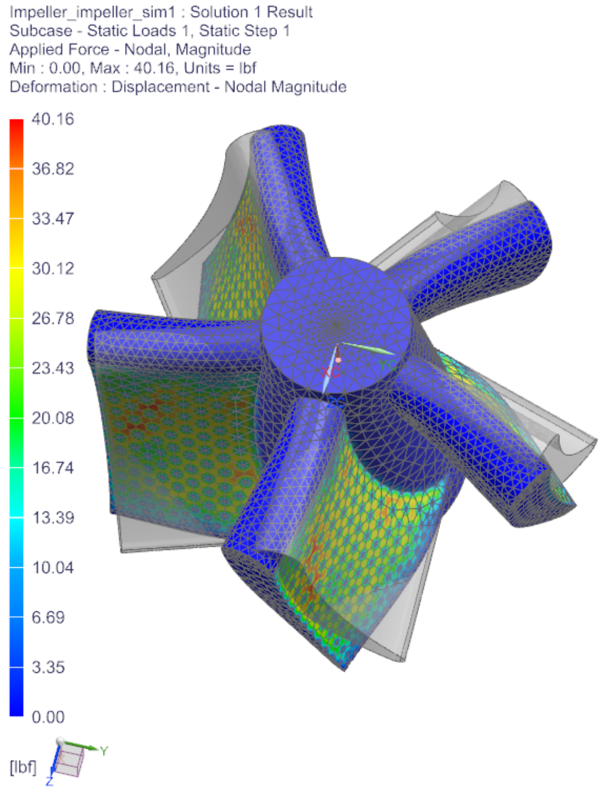
You can now see an animation of how the impeller is deformed as the loads are applied to the blades.

- To make any setting changes in the results display, click on the **Edit Post View** icon 

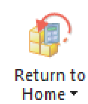
- Check the **Show undeformed model** and click **OK**



Now press on the **Play** button to see the animation. This will show the animation of deformation with the original shape in grey color, as shown in the figure below.



➢ Click on the **Stop** button to stop the animation

➢ Click on **Return to Home** to go back to the FEA model 

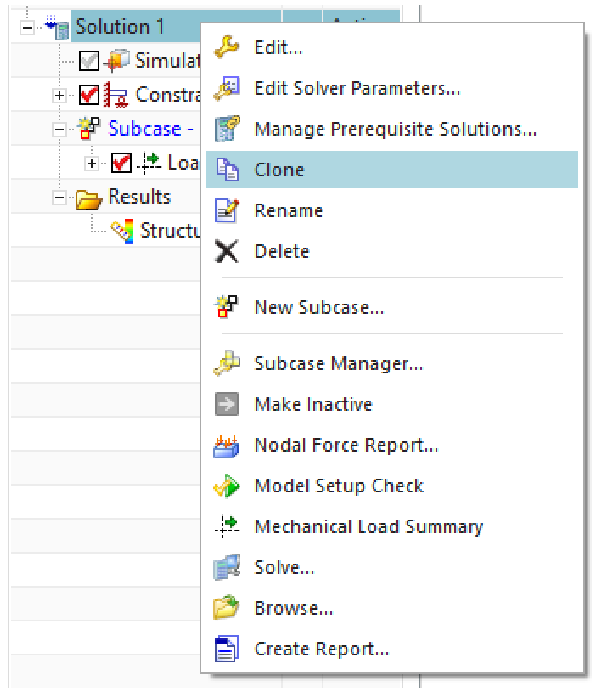
There are two ways to improve the accuracy of FEA results:

- Reduce the size of element
- Increase the order of interpolation polynomial (i.e. use quadratic or even cubic instead of linear polynomials)

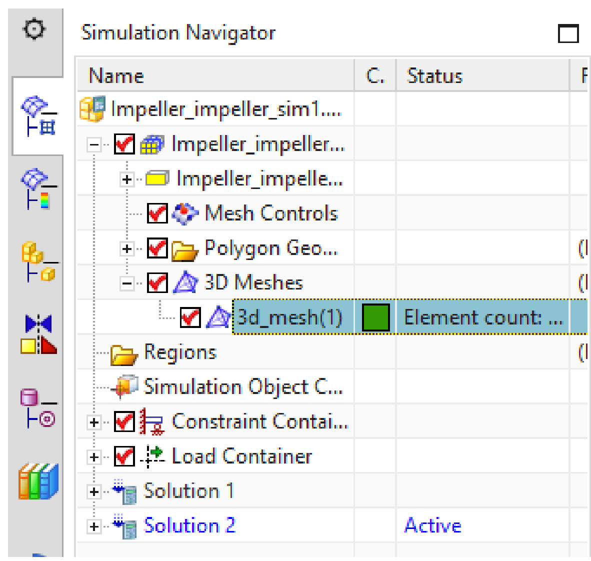
Let us try creating a simulation using different settings.

➢ Right-click on **Solution 1** in the **Simulation Navigator**

- Choose **Clone** to copy the first simulation



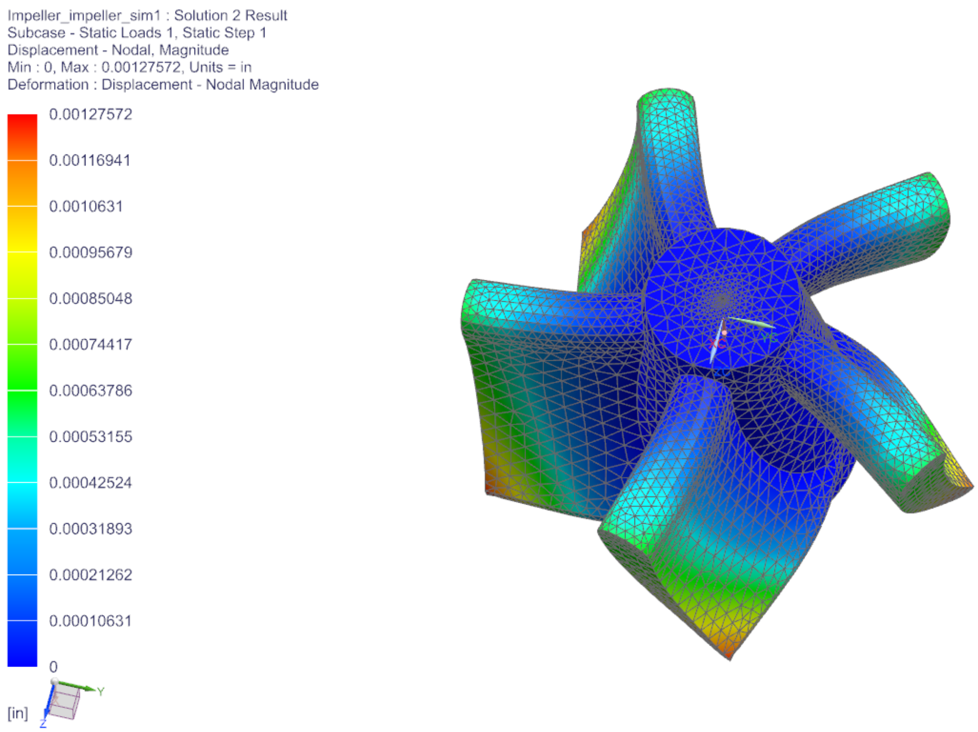
- Once **Copy of Solution 1** is created, rename it to **Solution 2**
- From the **Simulation Navigator**, under **3D Meshes**, right click on the **3D Mesh (1)** and click **Edit**
- In the popup dialog, change the **Type** to **TETRA4**
- Click **OK**
- Click on the **Solve** icon  to solve the simulation
- Click **OK**



The *Analysis Job Monitor* should show the status of *Solution 2* to be *Completed***.**

- Click **Cancel**
- In the **Simulation Navigator**, double-click on **Results for Solution 2**

The figure below shows the analysis. You can observe the change in the maximum value. Save all the simulations and close the files.

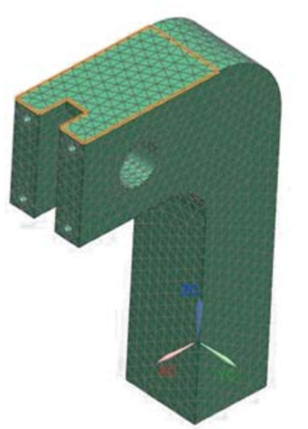


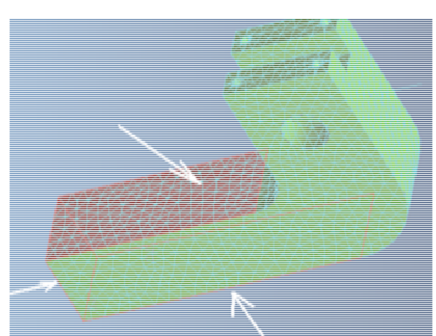
8.8 EXERCISES

8.8.1 Arbor Press Bar

Open the file '**Arborpress_L-bar.prt**' and do a similar structure analysis, considering the material as steel. For the mesh, the element size and type should be '10' and 'Tetra(10)'. For the loads, apply a normal pressure with a magnitude of 500 on the top surface as shown in the first figure below.

For the boundary conditions, fix the three flat faces (the front highlighted face, the face parallel to it at the backside and the bottom face) as marked in the second figure below.





8.8.2 Rocker Arm

In this exercise, you will examine the effect of element type and mesh size on the results of finite element analysis. Open the Rocker Arm you modeled in chapter 4. Assign the following material properties: Young's modulus = 3.0×10^7 psi, Poisson's ratio = 0.29, and mass density = 7.35×10^{-4} slug/in^3. Fix both the counter bore hole and the counter sunk hole (i.e. fix the cylindrical faces) as shown in the figure below and apply a pressure load of magnitude 600 psi normal to the face shown. For each of the four following cases, obtain the deflection contour and Von-Mises stress contour.

a) **Tetra 4** elements, element size – **0.2**
b) **Tetra 4** elements, element size – **0.05**
c) **Tetra 10** elements, element size – **0.2**
d) **Tetra 10** elements, element size – **0.05**

Please answer the following questions: What are the maximum deflection and the maximum stress for each of the four cases? Which of the four cases provides the most accurate results? How are the FEA results affected by the type and size of elements?