# **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**



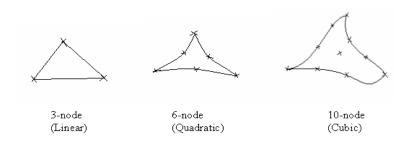
2-node (Linear)

3-node (Quadratic)

4-node (Cubic)

### Two-dimensional elements

Triangular:

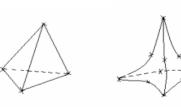


Quadrilateral:



Three-dimensional elements

Tetrahedral (a solid with 4 triangular faces):





10-node (Quadratic)

20-node (Cubic)

### Hexahedral (a solid with 6 quadrilateral faces):



8-node (Linear)



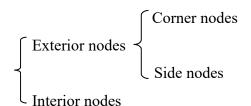


32-node (Cubic)

20-node

(Quadratic)

### Types of 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

<u>Starting the Simulation</u>: 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.

<u>Applying the Boundary Conditions</u>: 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.

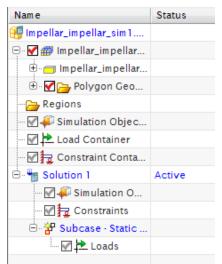
# **8.2 SIMULATION CREATION**

- Copy and paste the file Impeller\_impeller.prt into a new folder to avoid changes being made to the assembly
   You can download this file from: https://me5763.github.io/lab/assets/models/files\_ch8/Impeller\_impeller.prt
- > Click on New  $\rightarrow$  Simulations if the part is NOT already opened in the NX window

Machining Line Planner Man	ufacturing	Inspection	Mechatronics (	Concept Designer	Press Line	ine Designer	Ship Structures
Ship General Arrangement	Model	DMU	Drawing	Layout	Simulation	Additive	Manufacturing
lemplates				~	Preview		^
Filters				^			0
Relationship Stand-alone Pa	rt 🔻	Units	Inches	-			EW N
Name		Туре	Units	Relationship			Line
💮 NX Nastran		Fem	Inches	Stand-alo 🔨			
NX Nastran		Sim	Inches	Stand-alo		$\geq$	
MX Nastran Acoustic	1	Fem	Inches	Stand-alo			
🐠 NX Nastran Acoustic	:	Sim	Inches	Stand-alo			
🍘 NX Nastran Vibro-Acoustic	1	Fem	Inches	Stand-alo			
🐠 NX Nastran Vibro-Acoustic	:	Sim	Inches	Stand-alo			
🎒 NX Nastran Design	1	Fem	Inches	Stand-alo	Properties		^
🐠 NX Nastran Design		Sim	Inches	Stand-alo	Name: NX Nastra	an	
🎒 Samcef	1	Fem	Inches	Stand-alo	Type: Fem		
🐠 Samcef		Sim	Inches	Stand-alo	Units: Inches		
Simcenter Thermal/Flow	1	Fem	Inches	Stand-alo	Last Modified: 08	3/04/2017 12:4	2 PM
🐠 Simcenter Thermal/Flow		Sim	Inches	Stand-alo 🗸	Description: Emp	tv NX Nastrar	Fem
<				>		,	
ew File Name							^
Name fem1.fem							
in the second second							
Folder C:\Program Files\Siem	nens\NX 12.0	UGII			2		
Part to reference							^

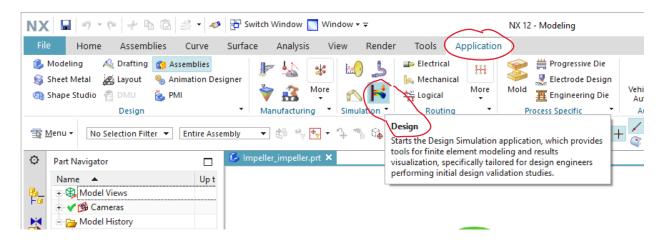
164

> Open this newly copied file



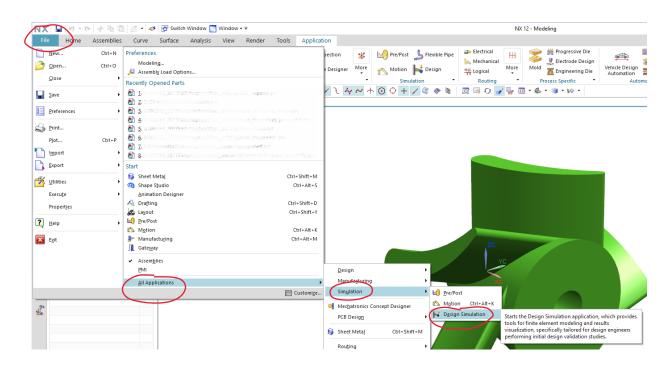
> If the part is already opened in NX, then from the top ribbon bar, click on Application  $\rightarrow$ 

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.

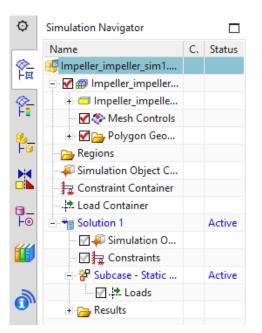
The default Solver type is NX Nastran Design and Analysis type is Structural.

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.

New FEM a	nd Simulation	ਹ X	😳 Solutio	
File Names		^	Solution	
Impeller_impell	er_fem1.fem		Name	Solution 1
Impeller_impell	er_sim1.sim		Solver	NX Nastra
CAD Part		^	Analysis Ty	/pe Structural
Associate to	Master Dart		Solution Ty	ype Linear Sta
Part Impeller_i			Autom	atically Create Ste
Idealized Part	inpener ·		Linear Sta	atics - Single Co
Create Idea	alized Part		General	Result Options
Name		^	Descriptio	
Impeller_im	peller_fem1_i.prt			ent Iterative Solve
			Max Job 1	
Bodies		^	🗌 Inertia	Relief
Bodies to Use	All Visible	•	Bulk D	Data Listing
Polygon Body	Resolution Standard	•	Flat Shell	Rz Stiffness Facto
Geometry	<u>-</u>	<u> </u>	Auton	matic Selection of
Geometry			Ignore	e Material Tempe
	Geometry Options		Bounda	ry Condition Cor
		_	🗌 Run Jo	ob in Foreground
Solver Enviror	iment	^		
Solver	NX Nastran Design	•		
Analysis Type	Structural	•		
Description		^		
	ОК	Cancel		

Solution						<del>ں</del>	X
Solution							۸
Name	Solution 1						
Solver	NX Nastrar	n Design					•
Analysis Type	Structural						•
Solution Type	Linear Stati	ics - Single	Constrair	nt			•
Automatically	/ Create Step	or Subcas	e				
Linear Statics -	Single Cor	nstraint					۸
General Resu	It Options	Contact					
Description							
Element Iter	ative Solver				 		-
Max Job Time							7
🗌 Inertia Relief							
Bulk Data Lis	sting						
Flat Shell Rz Stif	fness Factor	100					•
Automatic S	election of [	Dependent	DOFs				
Ignore Mate	rial Tempera	ture Depe	ndence				
Boundary Cor		rol Variable	5			Ň	·
Run Job in F	oreground						
				OK			
				OK	Apply	Cancel	

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  $30 \times 10^6$ )

> Choose **OK** to exit the **Isotropic Material** window

Now we have assigned the material to the impeller model.

ect Body Select Body (0)			_ ф
Science body (0)			Ψ
aterial List			^
🕉 Library Materials			-
ibraries			v
/laterials			^
Name	Used	Catego	ry
Copper_C10100		METAL	-
Engine_Oil_Liquid		OTHER	
Epoxy		OTHER	
Ethylene_Glycol_Liquid		OTHER	
Freon_Liquid_R12		OTHER	
Glycerin_Liquid		OTHER	
Helium_Gas		OTHER	
Hydrogen_Gas_H2		OTHER	
Inconel_718-Aged		METAL	
Iron_40		METAL	
Iron_60		METAL	
Iron_Cast_G25		METAL	
Iron_Cast_G40		METAL	~
<		>	
lew Material			^
ype Isoti	ropic		-
		1	
Create		8	

Isotropic Material			ა x
Property View			^
All Properties			•
Name - Description			^
Steel			
Label			1
Description			v
Categorization			v
Properties	-		^
Mass Density (RHO) 0.284	4		lbm/in³ ▾ 〓
	Elastic Constants		^ ^
Strength	Young's Modulus (E)	30e6	lbf/in <sup>2</sup> • =
Durability	Tourig's Modulus (E)	5000	lbf/in² ▼ =
Formability	Major Poisson's Ratio	$\sim$	•
Thermal/Electrical	Poisson's Ratio (NU)	0.27	=
Creep	Shear Modulus (G)		lbf/in <sup>2</sup> • =
Viscoelasticity			
	Structural Damping Coefficient (GE)		=

### **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.

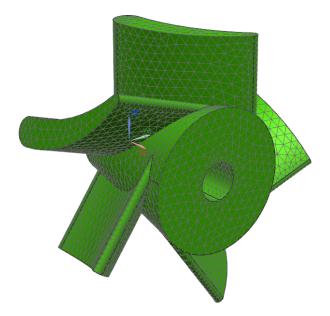
Then, there are two types of *Tetrahedral Elements* available in NX 12. One is 4-node and the other is 10-node.

- 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 tradeoff you need to consider. If you choose a smaller 3D Tetrahedral Mesh υx Objects to Mesh Λ Select Bodies (1) Element Properties ^ A CTETRA(10) • Type Mesh Parameters Λ Element Size 1 in - -9 Attempt Free Mapped Meshing Attempt Multi-Block Cylinders Mesh Quality Options Δ Mixed -Midnode Method Geometry Tolerance Jacobian 10 Ŧ Mesh Settings ^ Surface Curvature Based Size Variation 50.0000 Internal Mesh Gradation 1.0500 Target Internal Edge Length Limit Minimum Two Elements Through Thickness Model Cleanup Options ۸ Small Feature Tolerance (% of Element Size) 10.0000 Minimum Element Length (Read-Only) 0.1 8 Preview Δ Boundary Nodes Apply Cancel

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.

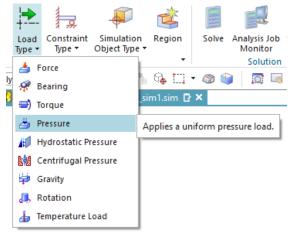


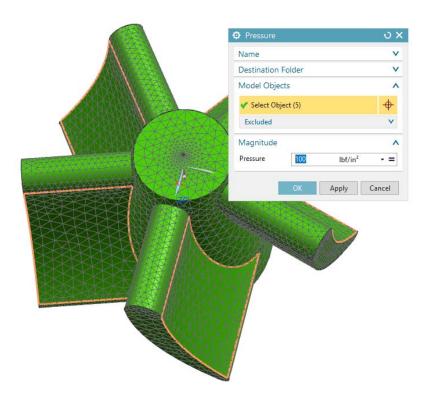
# 8.5 LOADS

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<sup>2</sup> 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<sup>2</sup> (psi)

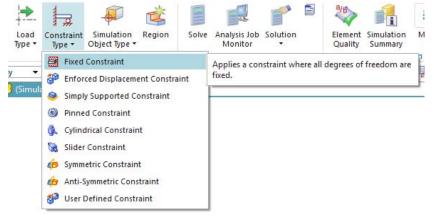




# **8.6 BOUNDARY CONDITIONS**

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 FixedConstraint

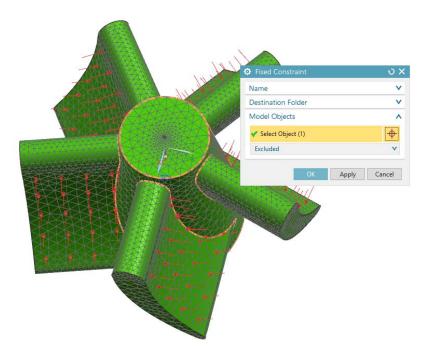


This type of constraint will restrict the selected entity in six

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

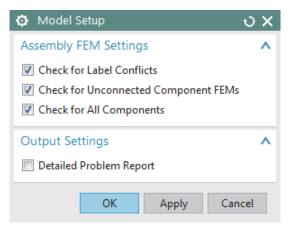
➢ Click on the Menu → Analysis → Finite Element Mode Check → Model Setup or

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.



🗘 Information ? _ 🗆 🗙
🔊 🕪 🖻 🖻 × 🕈 🧒 🗖 -
Boundary Condition Geometry Associativity Check Summary
The following Boundary Conditions are applied to geometry having partial association of nodes and elements. Fixed(1) This may lead to unexpected solution results. Use Boundary Condition expanded display mode to show the distribution from geometry to the finite element model.
Mesh-Based Errors Summary
Total: 0 errors and 0 warnings
Material-Based Errors Summary
Total: 0 errors and 0 warnings
Solution-Based Errors Summary
Iterative Solver Option More than 80 percent of the elements in this model are 3D elements. It is therefore recommended that you turn ON the Element Iterative Solver in the "Edit Solution" dialog.
Total: 0 errors and 0 warnings
Load/BC-Based Errors Summary
Total: 0 errors and 0 warnings
Nastran Model Setup Check completed

Click on the Solve icon

This will open the Solve window.

> Click **OK** without making any changes

Solve	υx
Solving Options	^
Submit Solve	-
Model Setup Check	
Resolve Label Conflicts In Sim Part (None)	
Edit Solution Attributes	
Edit Solver Parameters	
Prerequisite Solutions Chain	v
OK Car	ncel

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

Ō

8

-0

ര

ø

Simulation Navigator

🐠 Impeller\_impeller\_sim1....

🖃 🗹 💮 Impeller\_impeller...

🗄 🗇 Impeller\_impelle...

🗄 🗹 👝 Polygon Geo...

Simulation Object C...

🛛 🛃 🔁 Constraint Contai...

🗑 🐗 Simulation O...

😵 Subcase - Static ...

≪ Structural

🖌 🛃 Load Container

🗄 🗹 🚖 Constraints

🗄 🖌 📩 Loads

👝 Results

🗄 🗹 🖄 3D Meshes

👝 Regions

Solution 1

🖌 📀 Mesh Controls

Name

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

C. Status

Active

Active

Inferred

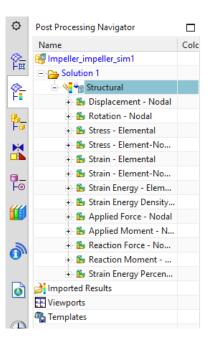
174

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.

Reset List
Analysis Job Information
Check Analysis Quality
Cancel

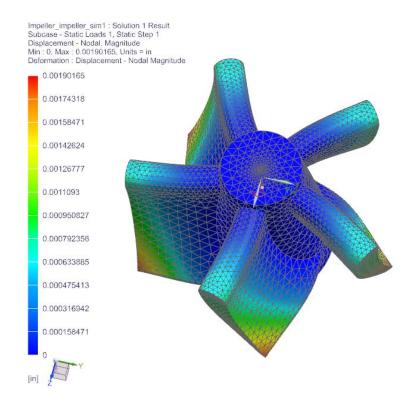
Completed - impeller\_impeller\_sim1-solution\_1

Analysis Job Monitor

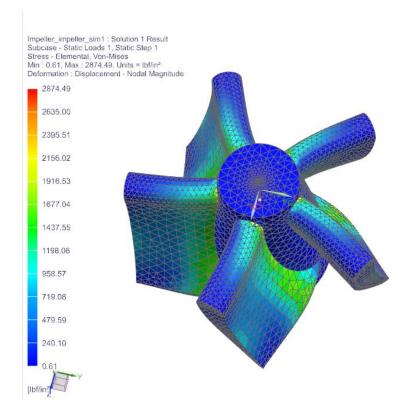


#### Double-click on the Displacement-Nodal

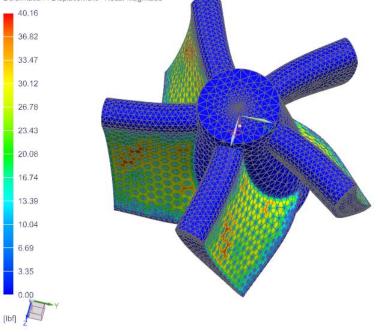
The screen will now appear as shown below. You can easily interpret the results from the colorcoding. The orange-red color shows the maximum deformation zones and the blue area shows the minimum deformation zones. You can observe that because the conical core is fixed, it experiences zero deformation.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

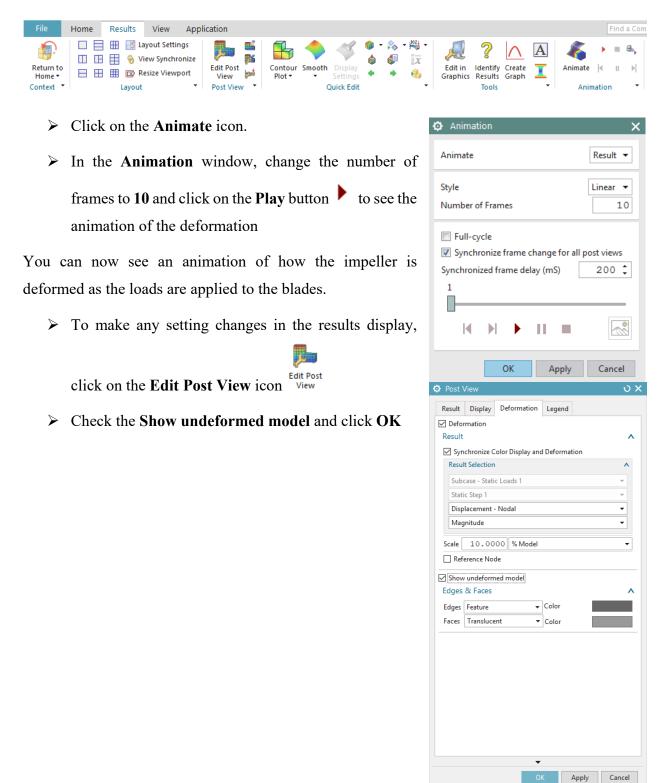


Impeller\_impeller\_sim1 : Solution 1 Result Subcase - Static Loads 1, Static Step 1 Applied Force - Nodal, Magnitude Min : 0.00, Max : 40.16, Units = Ibf Deformation : Displacement - Nodal Magnitude

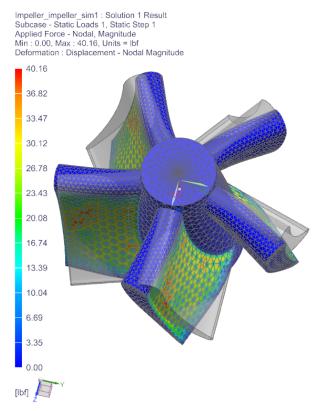


### 8.7.3 Simulation and Animation

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



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

Click OK

rename it to Solution 2

(1) and click Edit

**TETRA4** 

Click on the Solve icon to solve the simulation

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

> Once Copy of Solution 1 is created,

> From the Simulation Navigator, under

> In the popup dialog, change the **Type** to

3D Meshes, right click on the 3D Mesh

Click OK

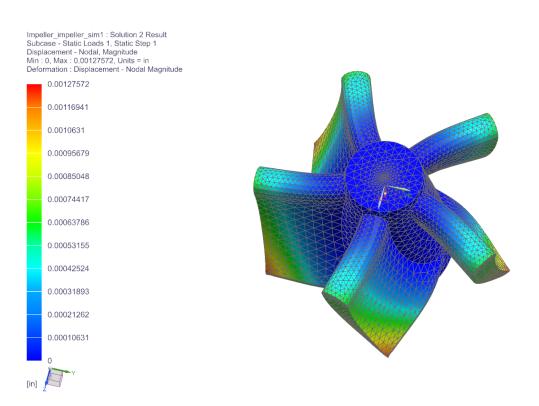
The *Analysis Job Monitor* should show the status of *Solution 2* to be *Completed*.

- 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.

Solution 1		
- Sinulat	Ļ	Edit
🗉 🗹 🏣 Constra	<u>,</u>	Edit Solver Parameters
Bubcase -	7	Manage Prerequisite Solutions
🗄 🗹 📩 Loa	Pa	Clone
🖃 🦳 Results	Ľ	Rename
	×	Delete
	容	New Subcase
	ቃ	Subcase Manager
	⇒	Make Inactive
	<u>**</u>	Nodal Force Report
		Model Setup Check
		Mechanical Load Summary
		Solve
	2	Browse
		Create Report
L	_	

¢	Simulation Navigator	[		
	Name	С.	Status	F
- Ⅲ	🐠 Impeller_impeller_sim1			
ΓA	🖃 🗹 🍘 Impeller_impeller			
8_	🗉 🗇 Impeller_impelle			
F.	🛛 🗹 🐡 Mesh Controls			
<b>₿_</b>	🛨 🗹 🚘 Polygon Geo			(
Fø	🖃 🗹 🖄 3D Meshes			(
	🖌 🛃 🐼 3d_mesh(1)		Element count:	
	- 🕞 Regions			(
•	- 🐺 Simulation Object C			
F⊚	🛨 🗹 🚘 Constraint Contai			
	🗉 🗹 보 Load Container			
	🛨 📲 Solution 1			
	🕀 📲 Solution 2		Active	
-				



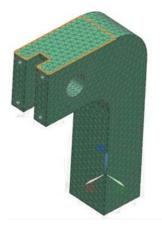
### **8.8 EXERCISES**

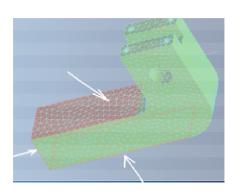
8.8.1 Arbor Press Bar

You can download this file from: ttps://me5763.github.io/lab/assets/models/files\_ch8/Arborpress\_Lbar.prt

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 =  $3.0 \times 10^7$  psi, Poisson's ratio = 0.29, and mass density =  $7.35 \times 10^{-4}$  slug/in<sup>3</sup>. 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?

