MFG319544-L

Up and Running with Autodesk Inventor Nastran

Wasim Younis
Symetri

Learning Objectives

- Setup and run simulation using Inventor Nastran
- Interpret results using Inventor Nastran
- Apply industry best practices
- Apply top Inventor Nastran In-CAD tips

Description

Today Autodesk Inventor Nastran software is available as part of the Product Design & Manufacturing Collection, meaning many designers and engineers like you have the capability to adopt simulation early in the design process as this is where it has the most impact. Despite software availability and simulation continually becoming easier to use, there remains a fundamental question on the mind of every designer and engineer: How do I know my results are correct? This is the main reason for the lack of Autodesk Inventor Nastran adoption within workplace. Autodesk Inventor Nastran, although easy to use, has a comprehensive set of tools to help you answer this question and make sense of your results. These include mesh convergence, numerous result plots, section views, and much more. The session will include workflows and tips based on industry best practices. The goal of this session is to make you more confident in using Autodesk Inventor Nastran software, and thus help you make better and more-innovative products within your workplace.

Speaker

An Autodesk simulation solutions manager with more than 30 years of experience in the manufacturing field, including working at Rolls Royce and British Aerospace. Has been involved with Autodesk simulation software from when it was first introduced, and is well-known throughout the Autodesk simulation community, worldwide.

He has also authored the Up and Running with Autodesk Inventor Professional and Inventor Nastran books. He also runs a dedicated forum for Autodesk simulation users on LinkedIn – Up and Running with Autodesk Simulation. Currently he is employed @ Symetri (http://www.symetri.com) – an Autodesk value added services partner across UK and Northern Europe.

<u>Contact Details;</u>
wasim.younis@symetri.com
https://www.linkedin.com/in/wasimyounis/
+44 (0) 7980 735244

Page intentionally left blank

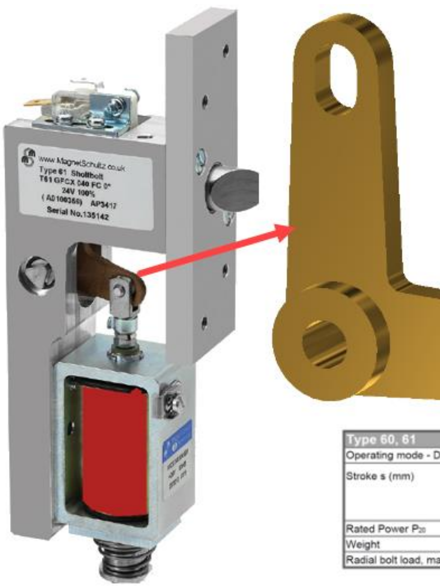
PART ANALYSIS EXAMPLE

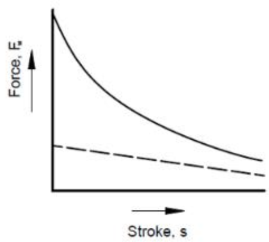
Key features and workflows introduced in this design problem.

	Key Features/Workflows
1	Pin Constraint
2	Bearing Load
3	Basic Mesh Settings
4	Global and Local Face Mesh Control
5	Results Convergence - Mesh Sensitivity Study
6	Safety Factor Result Plots
7	Redesign

Introduction

Magnet Schultz Ltd are the leading solenoid specialists in UK since 1967 and have high profile customers in a variety of industries including defence and automotive security. Typical products designed by Magnet Schultz include the T61 Right-angle Solenoid Shot bolts as shown below.





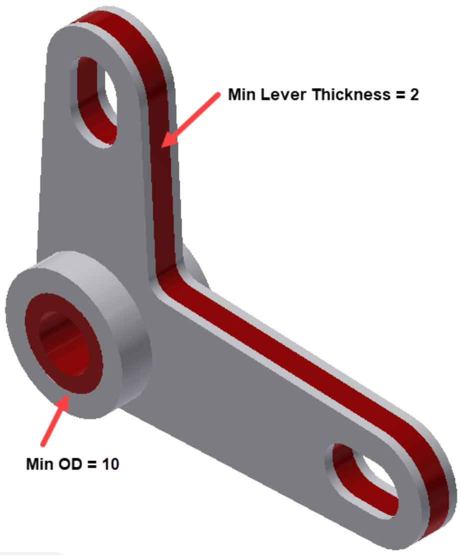
Type 60, 61			
Operating mode - Duty Rating ED		S1 100%	
Stroke s (mm)		Magnetic force F_M (N)	Spring return-force (N)
	0	30	9
	15	3.3	2.8
Rated Power P_{20}	(W)	14	
Weight	(kg)	1	
Radial bolt load, max allowable	(N)	3000*	

In this design problem we are going to analyse the lever using the following design information and goal.

Design Information

Material of Lever - **Steel**
Density - **7850kg/m³**
Youngs Modulus - **200GPa**
Poisson's Ratio - **0.29**
Yield Limit - **200MPa**
Load - **30N**
Minimum Safety Factor - **5**

Design Goal
Determine the minimum safety factor of the current design



Workflow of Exercise 1

IDEALIZATION
1 - Model geometry is already simple.

APPLY BOUNDARY CONDITIONS
1 - Apply materials, loads and constraints to simulate reality.
2 - Apply initial mesh settings.

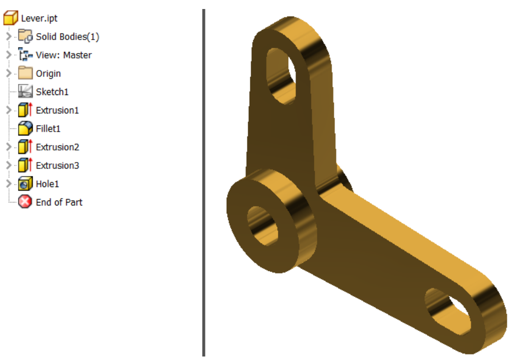
RUN SIMULATION AND ANALYSE
1 - Analyse and interpret results.
2 - Apply mesh control and achieve results convergence.

REDESIGN
1 - None

Idealization

The part file is already simple and therefore does not need to go through any further idealization.

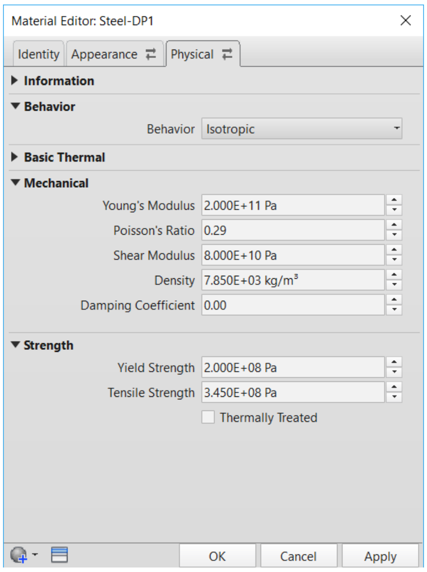
1. Open *Lever.ipt*.



2. Apply a new material to the lever with the following properties.

Youngs Modulus - **200GPa**
Poisson's Ratio - **0.29**
Density - **7850kg/m^3**
Yield Limit - **200MPa**

Select Mild Steel and then modify the individual properties.



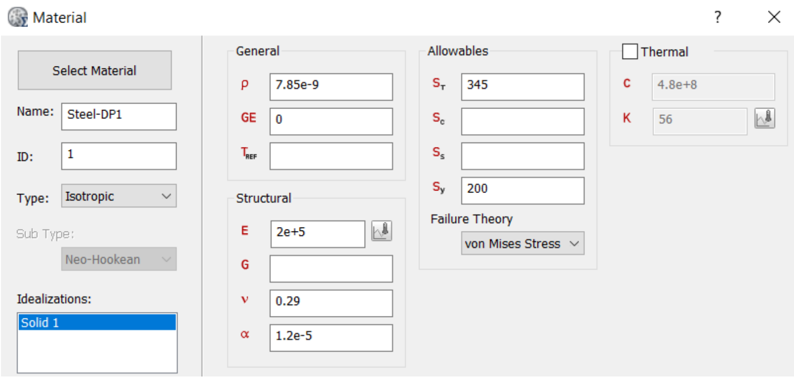
Shear Modulus value is not required by Inventor Nastran, as it calculates the value using existing material data. Inventor Nastran will automatically remove this value.

3. Rename Material name to **Steel-DP1** > Click **OK.**

Boundary conditions

4. Select **Environments** tab > Select **Autodesk Inventor Nastran**.

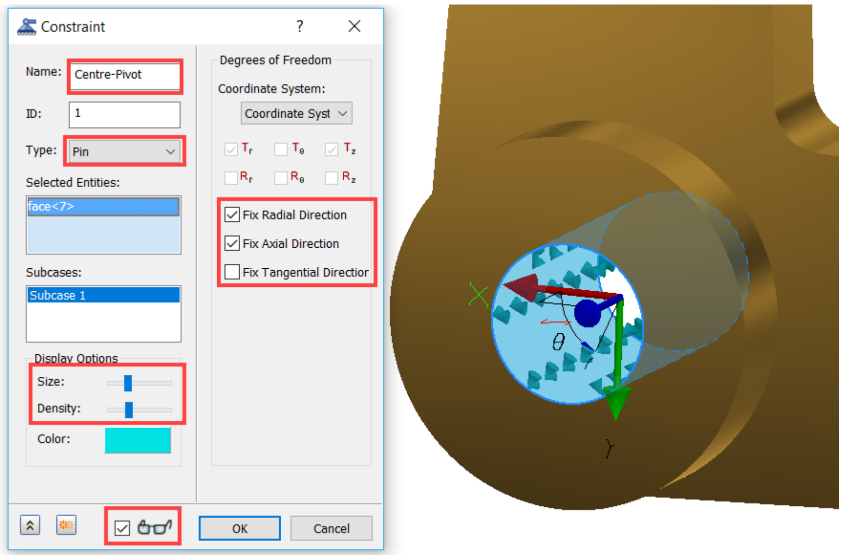
5. Double Click **Steel-DP1** under Materials node in the Model tree.



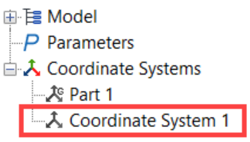
You can see the G field is blanked out. Here you can amend or add further values as required. Any additional data defined here will be saved only within the part file.

You can save new materials within Inventor Nastran that can then be accessed from other files.

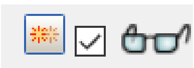
6. Click **OK** > Select **Pin Constraints** > Specify **Centre–Pivot** for Name > Select highlighted cylindrical face to apply constraint > Select both **Fix Radial Direction** and **Fix Axial Direction** > Select **Preview** so you can adjust display options as desired.



Applying a pin constraint will automatically create a customised cylindrical coordinate system with reference to the selected cylindrical face.

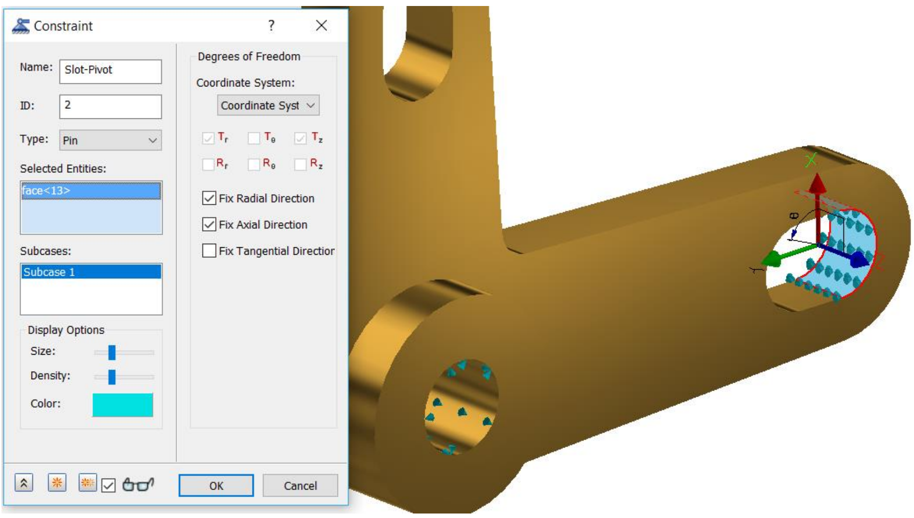


7. Select **Duplicate**.

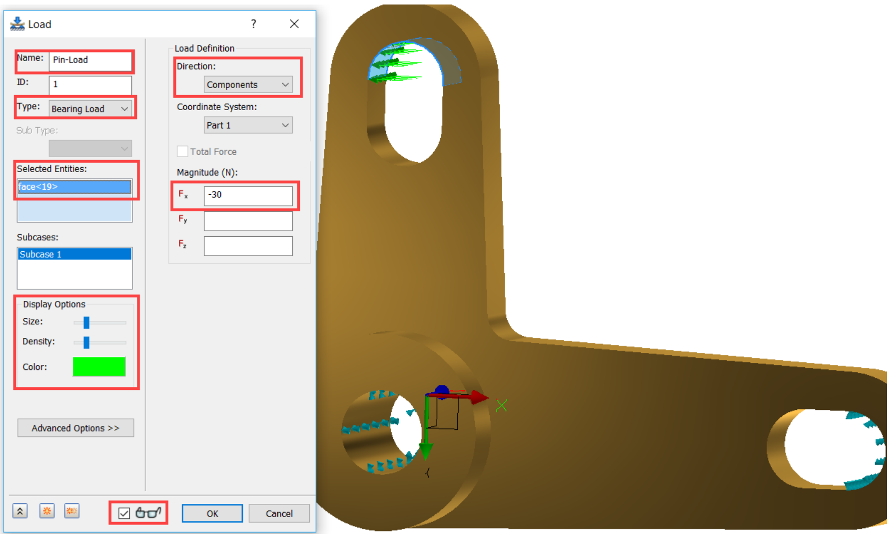


Centre–Pivot constraint has been created and now you can define a new constraint using same settings.

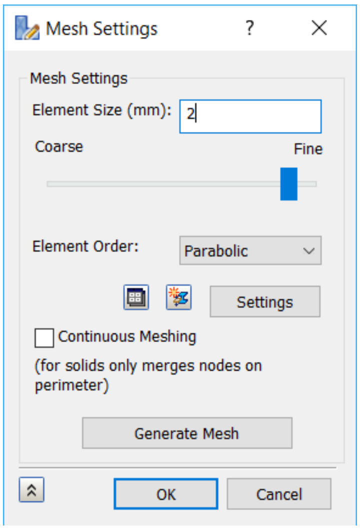
8. **Right click** within the Selected Entities box > Select **Clear All** to remove current selection > Select highlighted cylindrical face to apply new constraint > Specify **Slot–Pivot** for Name.



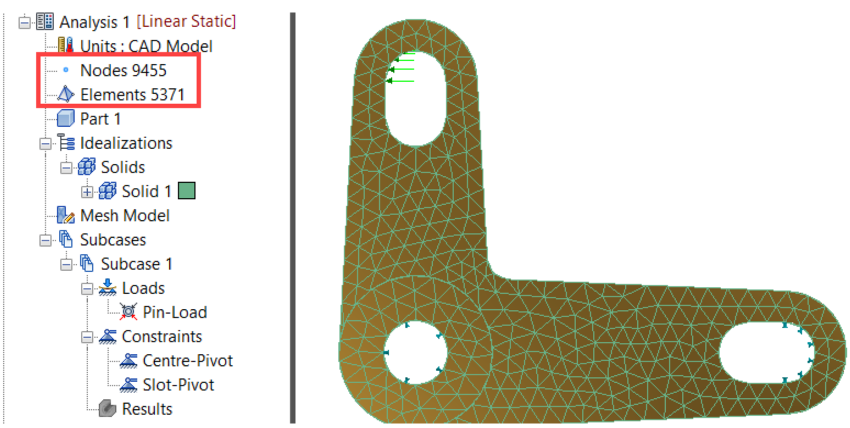
9. Click **OK** > Select **Loads** > Specify **Pin-Load** for Name > Select **Bearing Load** for Load Type > Select **Components** for Direction > Select highlighted cylindrical face to apply constraint > Specify **-30** for Magnitude (N) in Fx field > **Select Preview** so you can adjust display options as desired.



10. Click **OK** > Select **Mesh Settings** > Specify **2** for Element Size (mm).



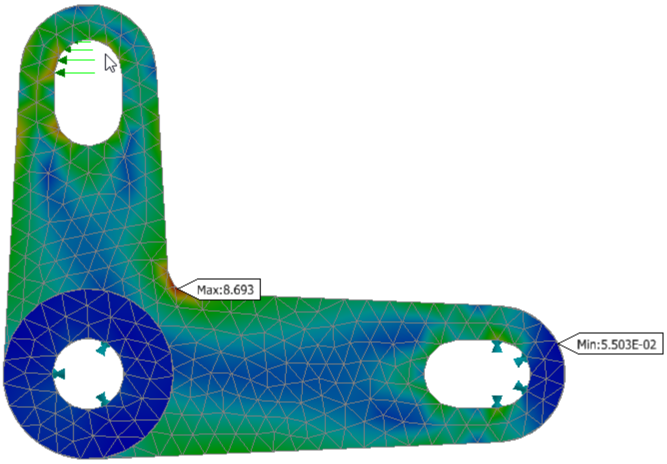
11. Click **OK**.



A total of 5371 tetrahedral elements will be generated.

Run simulation and analyse

12. Select **Run** > Click **OK** when run is complete.



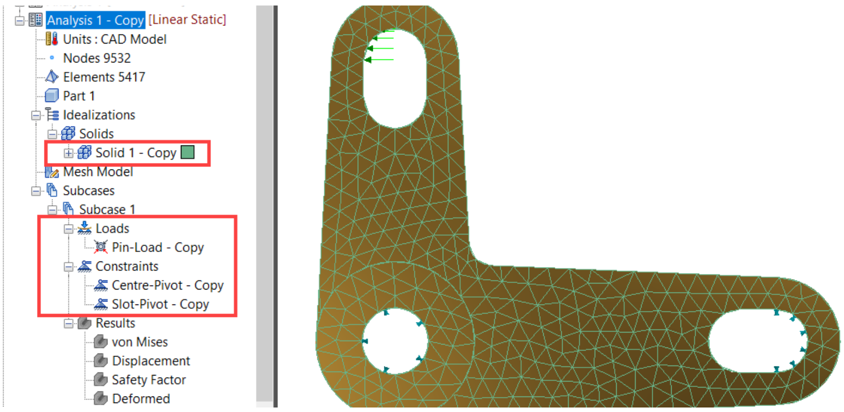
Once complete you will get several warnings. These warnings are referring to the tetrahedral element warnings. These warnings can be reduced by further refining the mesh.

The maximum von Mises stress is 8.693MPa giving us a factor of safety of 23.

$$\textbf{\textit{Factor of Safety}} = \frac{200}{8.693} = 23$$

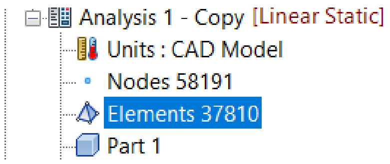
This suggests our design is considerably over-engineered meaning we can further optimise the design. Before this we need to check whether the stress value is sensitive to mesh changes.

13. Right click **Analysis 1** > Select **Duplicate**. This will duplicate idealizations, loads, and constraints.



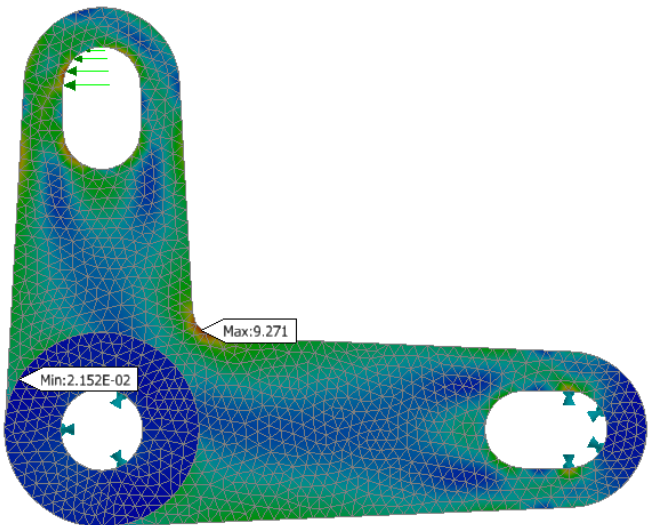
An alternative method is to create a new analysis and then drag and drop boundary conditions and other settings from the model tree to the analysis tree.

14. Select **Mesh Settings** > Specify **1** for Element Size (mm) > Click **OK**.



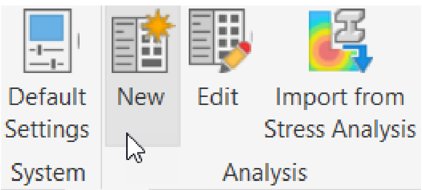
This will create 37810 elements. This is more than 7 times the first analysis. So, reducing the mesh by half does not necessarily produce twice as much elements.

15. Select **Run** again > Click **OK** when run is complete.



The maximum von Mises stress has increased to 9.271N that is about 7% increase in value. We will run the analysis again and see if this percentage difference reduces.

16. Select **New** Analysis from the Analysis panel > Click **OK**.

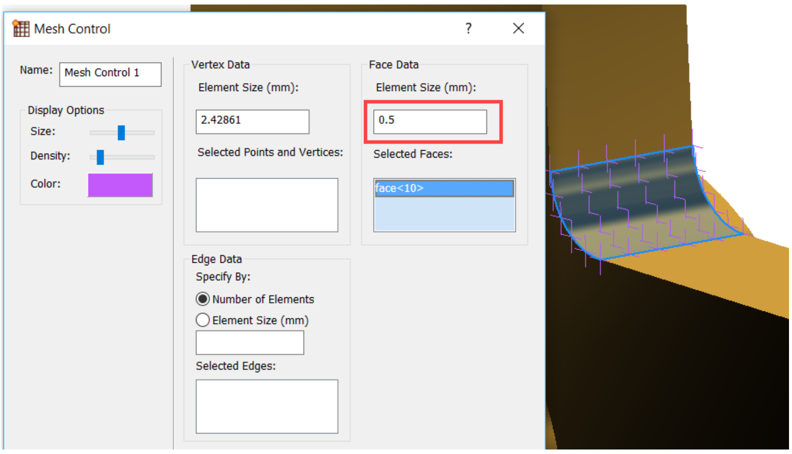


17. Select **Solid1** Idealization in the Model Tree > Keeping **left mouse button pressed** drag the **Solid1** idealization onto the Idealization node in the new Analysis 3 > Now **release left mouse button**. This will copy Solid1 idealization.

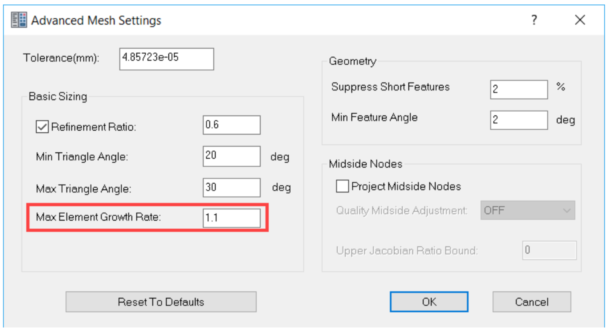
18. Repeat step 17 to copy **Centre-Pivot** constraint, **Slot-Pivot** constraint and **Pin-Load**. Making sure to release left mouse button on the Constraints node for constraints and Loads node for loads.

Now rather than changing the global mesh size we will define a local mesh control in the high stress area. The benefit of this method is that it will not generate excessive elements in low stress areas.

19. Select **Mesh Control** > Select Face Data option by clicking in the Selected Faces box > Specify **0.5** for Element Size (mm) > Select the highlighted face.



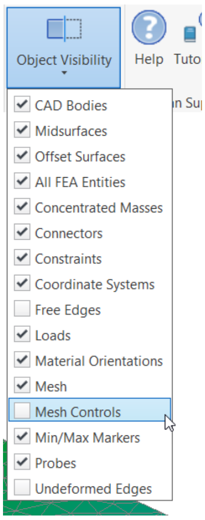
20. Click **OK** > Select **Mesh Settings** > Specify **1** for Element Size (mm) > Select **Settings** to access Advanced Mesh Settings > Specify **1.1** for Max Element Growth Rate.

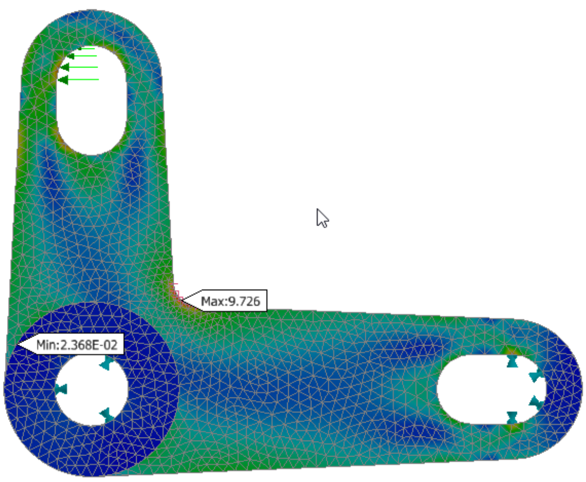


21. Click **OK** twice > Select **Run >** Click **OK** when run is complete.

22. Unselect **Deformed** from the results panel within the ribbon.

23. Select **Object Visibility** > Unselect **Mesh Controls**. This will hide symbols for mesh control.

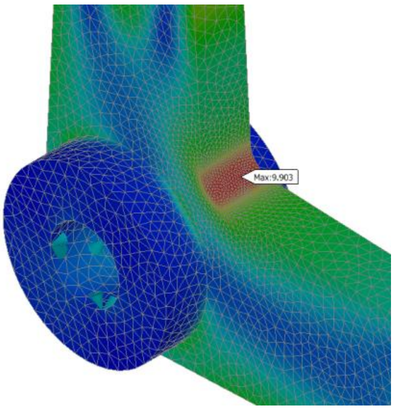




The maximum von Mises stress has now increased to 9.726N which is about 5% increase in value. The percentage change in maximum von Mises stress value has reduced and suggests the max value will eventually converge by further mesh refinement. At this stage we can take this value to calculate our safety factor or we can alternatively run one more simulation with a finer local mesh. We will use the latter option.

24. Right click **Mesh Control 1** > Select **Edit** > Specify **0.25** for Element Size > Click **OK** > Select **Generate Mesh**.

25. Select **Run** > Click **OK** when run is complete.



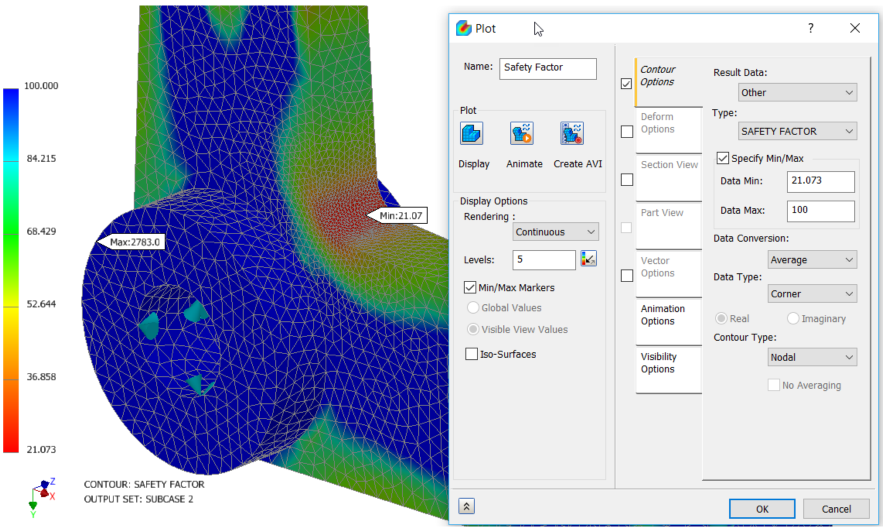
The maximum von Mises stress has increased to 9.903N which is about 1.8% increase in value from the 3rd analysis result. So, we can confidentially say the results have converged. We have a safety factor value of 20.

$$\boldsymbol{Factor\ of\ Safety} = \frac{200}{9.903} = 20$$

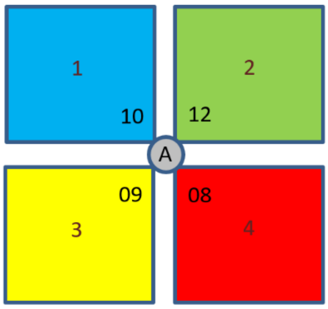
We can also plot Safety Factor plots in Inventor Nastran.

26. Double click **Safety Factor** results plot. To better visualise and understand safety factor results we can modify the colour legend scale.

27. Right click on **Safety Factor** Plot in browser > Select **Edit** > Select **Specify Min/Max** > Specify **100** for Data Max > Select **Display**.



The minimum safety factor, in same location, is 21.07 slightly higher than the previously calculated value. So, the question is why are the values different? The answer is that the calculated stress value is based on the average stress value at the node location. Whereas the safety factor plot values are based on taking the average of the nodal safety factors. This can be best explained by the following example. We will use 200MPa for yield limit for comparison purposes.



Safety factor calculation based on average stress at location A is 20.51.

Average Stress value = (10+12+9+8)/4 = 9.75

Safety Factor = 200/9.75 = 20.51

Safety factor calculation based on how Nastran In-CAD displays plots at Location A is 20.97.

Safety Factor for Element 1 = 200/10 = 20

Safety Factor for Element 2 = 200/12 = 16.667

Safety Factor for Element 3 = 200/9 = 22.222

Safety Factor for Element 4 = 200/8 = 25

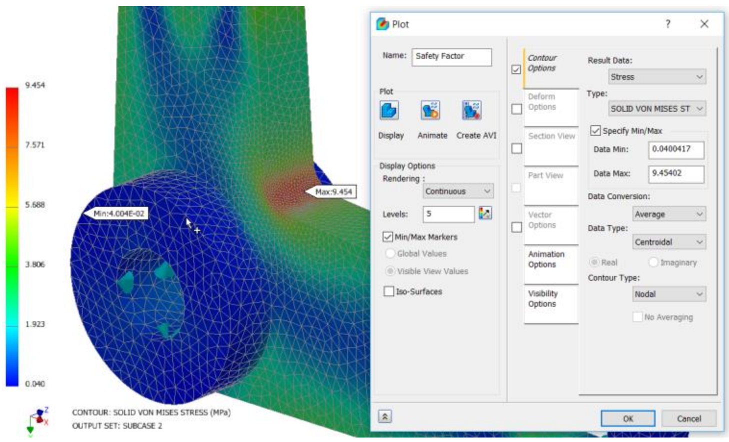
Average Safety Factor = (20+16.667+22.222+25)/4 = 20.97

So, this is the reason why there is slight difference in the safety factor results. Both results are correct as they have been calculated in slightly different ways. It is entirely up to yourself which value you take.

For comparison purposes changing the results to display the average centroidal results changes the stress value to 9.454MPa.

So, based on this value the factor of safety now becomes 21.15.

$$\boldsymbol{Factor\ of\ Safety} = \frac{200}{9.454} = 21.15$$



This is very close to the Safety Factor plot value of 21.07 with a difference of less than 0.5%.

Use average centroidal stress values if you would like to use safety factor plots.

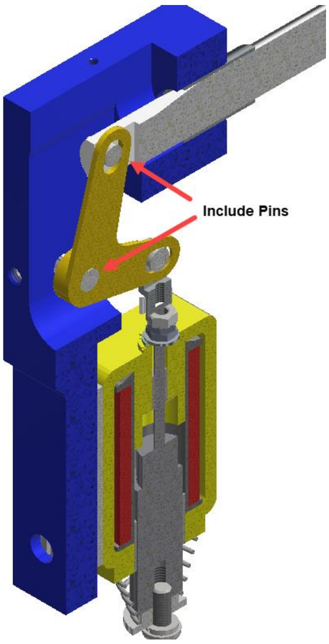
28. Save File **> Close** file.

ASSEMBLY ANALYSIS EXAMPLE

Key features and workflows introduced in this design problem

	Key Features/Workflows
1	Fixed Constraint
2	Frictionless Constraint
3	Bearing Load
4	Symmetry Conditions
5	Automatic Contacts
6	Advanced Mesh Settings
7	Global and Local Face Mesh Control
8	Display full model results from a half symmetry model

In this design problem we are going to analyse the lever as an assembly using the following design information and goal.



Design Information

Lever
Material of Lever - **Steel**
Density - **7850kg/m³**
Youngs Modulus - **200GPa**
Poisson's Ratio - **0.29**
Yield Limit - **200MPa**
Load - **30N**
Minimum Safety Factor - **5**

Pins
Material of Lever - **Steel Alloy**
Density - **7730kg/m³**
Youngs Modulus - **205GPa**
Poisson's Ratio - **0.3**
Yield Limit - **250MPa**
Load - **30N**
Minimum Safety Factor - **5**

Design Goal
Is to determine the minimum safety factor of lever.

Workflow of Exercise 2

IDEALIZATION
1 - Using half a model with symmetry boundary conditions.

BOUNDARY CONDITIONS
1 - Apply materials, loads and constraints to simulate reality.
2 - Apply mesh setting based on final settings off design problem 1.

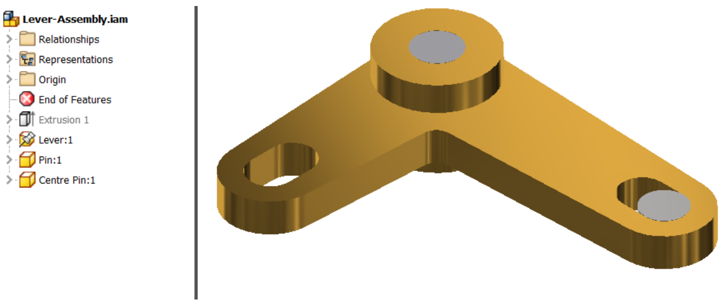
RUN SIMULATION AND ANALYSE
1 - Analyse and interpret results.
2 - Compare results with design problem 1.

REDESIGN
1 - None.

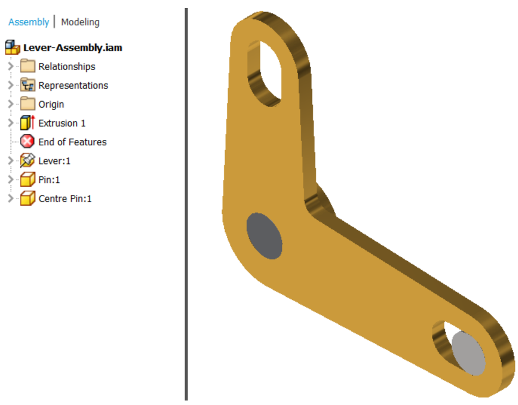
Idealization

We are going to analyse half of the assembly using symmetry conditions. This is a standard procedure when both geometry and loading are symmetrical. With the added advantage of reducing the file size by at least half with faster run times.

1. Open *Lever-Assembly.iam*.



2. Move **End of Features** below **Extrusion 1**.

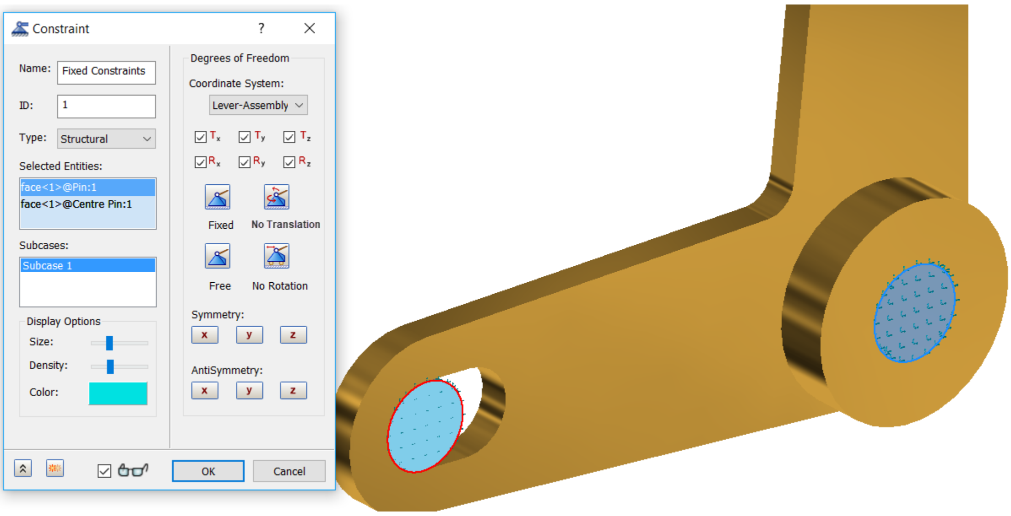


Extrusion 1 is defined in the assembly environment as it is easier to half all parts in one extrusion when compared to extruding in part environment for each component. The other advantage of this method is it does not alter the original parts.

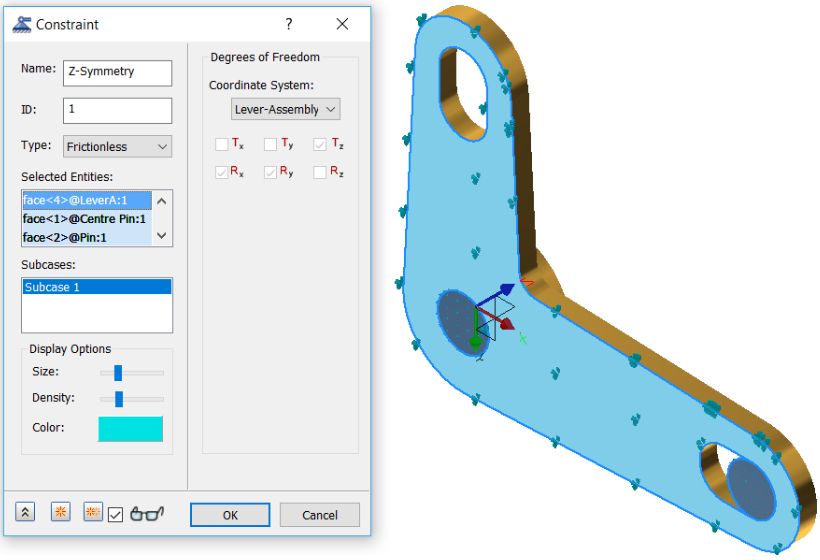
Boundary conditions

3. Select **Environments** tab > Select **Autodesk Inventor Nastran**.

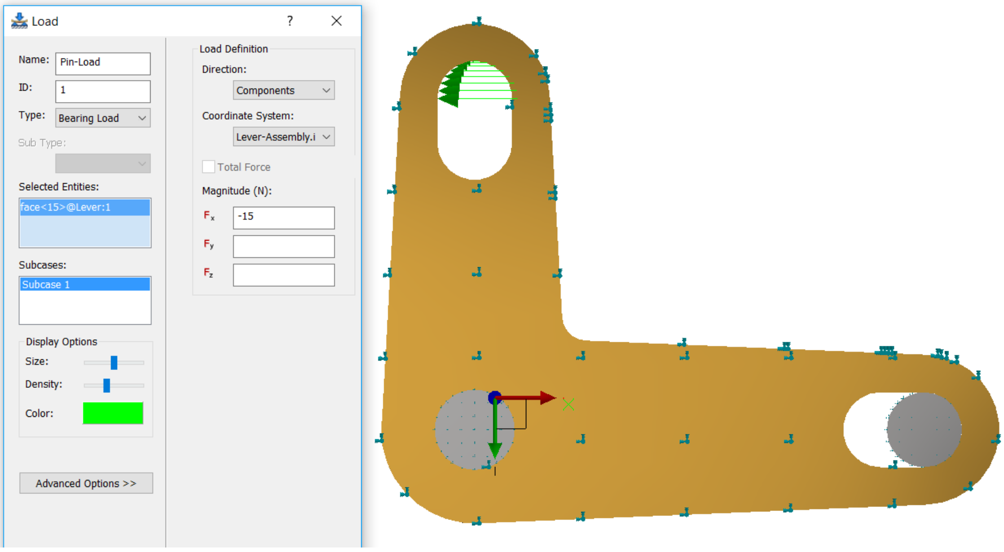
4. Select **Constraints** > Specify **Fixed Constraints** for Name > Select highlighted faces of both pins to apply constraints > Select **Preview** so you can adjust display options as desired.



5. Click **OK** > Select **Frictionless Constraints** > Select all 3 highlighted faces to apply new constraint > Specify **Z-Symmetry** for Name > Select **Preview** so you adjust display options as desired.

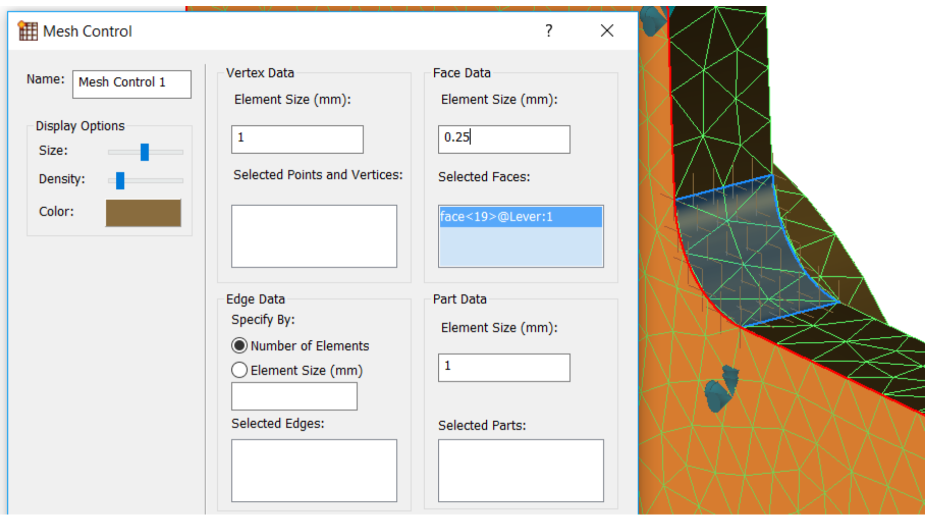


6. Click **OK** > Select **Loads** > Specify **Pin-Load** for Name > Select **Bearing Load** for Load Type > Select **Components** for Direction > Specify **-15** for Magnitude in Fx field > Select same face as exercise 1 > **Select Preview** so you can adjust display options as desired.



15N is specified as we are using half a model.

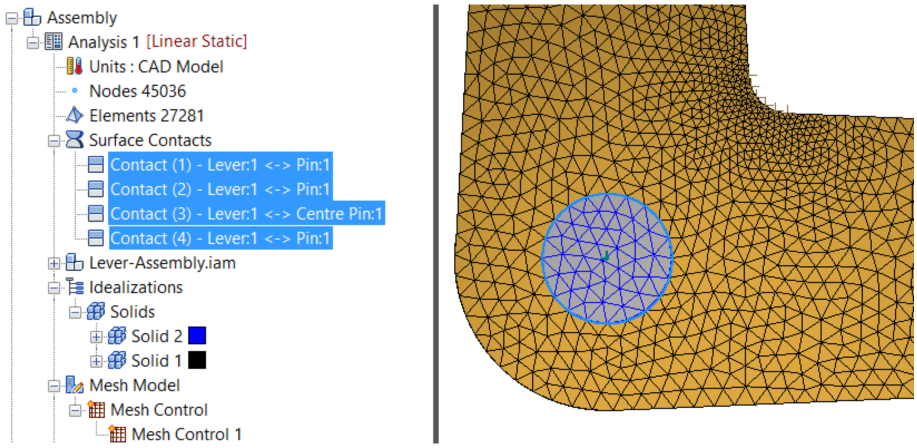
7. Click **OK** > Select **Mesh Control** > Select Face Data option by clicking in the Selected Faces box > Specify **0.25** for Element Size (mm) > Select the highlighted face.



8. Click **OK** > Select **Mesh Settings** > Specify **1** for Element Size (mm) > Select **Settings** to access Advanced Mesh Settings > Specify **1.1** for Max Element Growth Rate > **Click OK**.

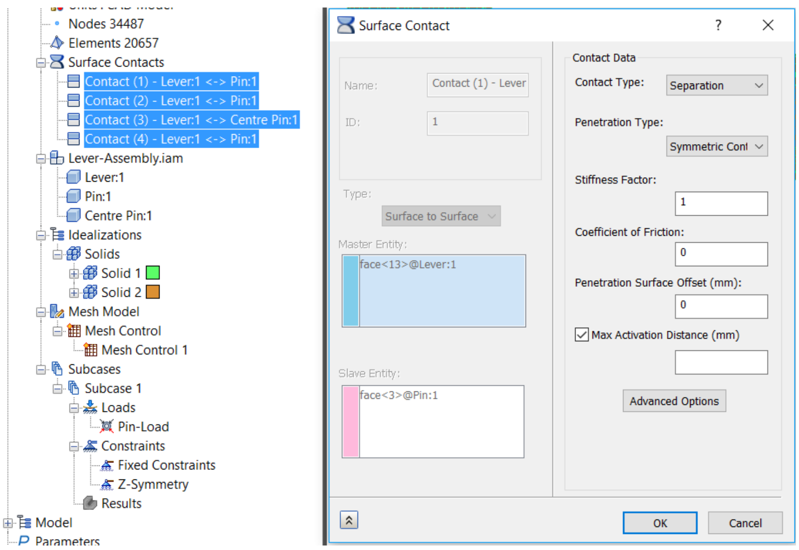
These mesh settings are same as the final mesh settings for the lever example in exercise 1 (Analysis 4).

9. Select **Automatic** contacts from the Contacts panel.



This will create 4 bonded contacts between lever and the two pins. Pins in reality, are not bonded and instead allow rotation and possible separation from the lever when a high enough load is applied.

10. Select all created contacts > **Right click** selected contacts > Select **Edit** > Select **Separation** for Contact Type.



11. Click **OK**.

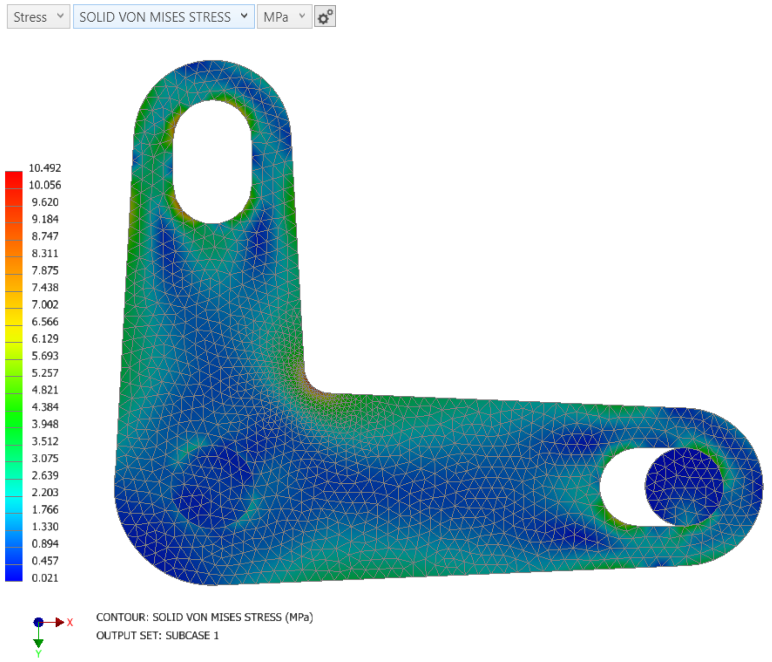
Run simulation and analyse

12. Select **Run** > Click **OK** once run is complete.

13. Select **Object Visibility** > Unselect **All FEA Entities** > Reselect **Mesh**.

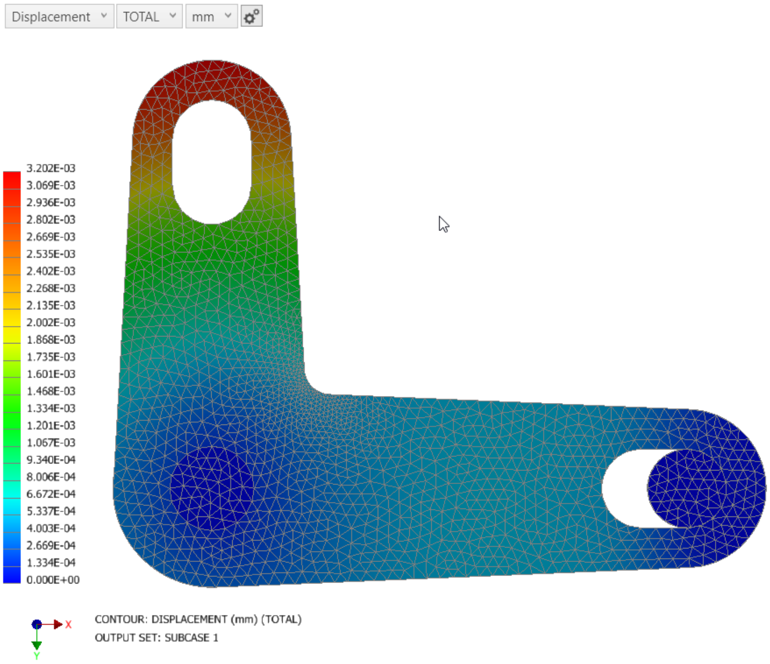
You may need to reselect and then unselect All FEA entities

14. Select **von Mises stress** plot > Unselect **Deformed** from the results panel within the ribbon.



The maximum stress (nodal average based) is 10.49MPa.

15. Select **Displacement** from the Results Navigation bar.

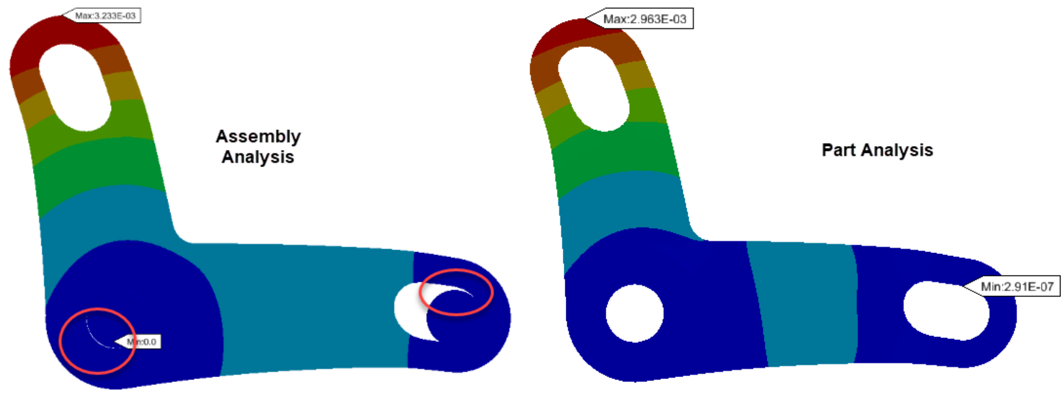


Maximum displacement is 0.0032mm

The maximum stress and displacement for the lever when analysed as a single part are;

- von Mises (nodal average based) is 9.903MPa.
- Max displacement is 0.00296mm.

The percentage difference between the stress results is 5.9%. So why is there a difference in the results? Should you analyse as a single part or as assembly? Let’s take a closer look.



We can see that the assembly analysis shows more movement which is due to the separation contacts allowing the lever to separate from pins, like reality. This behaviour cannot be simulated in part analysis. The extra movement in the assembly will result in a higher stress than the part analysis.

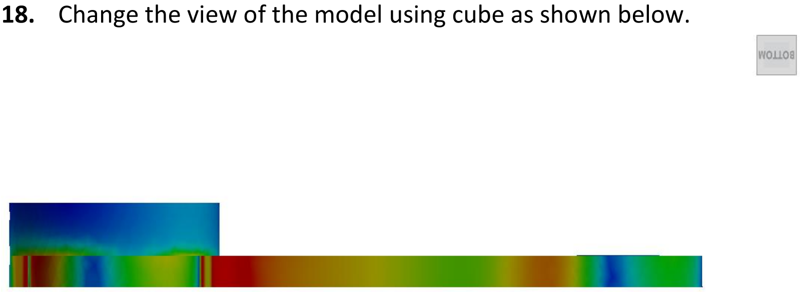
As the difference between both analyses are not significant it is entirely down to you of which method you prefer. It is also good practice to start from part analysis to get a good idea of the behaviour of the part and then move onto assembly analysis to get a more realistic behaviour.

Since we analysed a half model we can use ground planes to display complete model results.

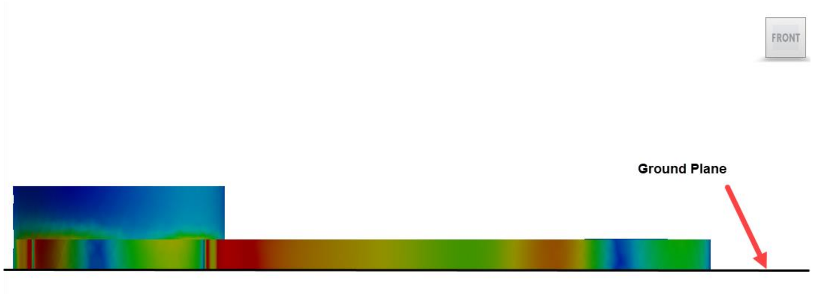
16. Unselect **Deformed** unless already unselected > Select **Object Visibility** > Unselect **Mesh** > Unselect **Min/Max Marker**.

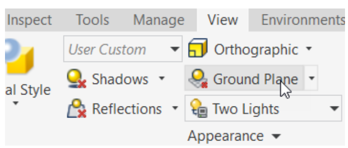
17. Right click **von Mises stress** results > Select **Edit** > Select **Stress Solid Von-Mises** results unless already selected > Select **Specify Min/Max** > Specify **0** for Data Min and **5** Data Max > Select **Display** > Click **OK**.

18. Change the view of the model using cube as shown below.

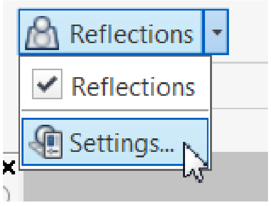


19. Right click cube > Select **Set Current View** as **Front** > Select **View** tab > Click on **Ground Plane** icon in Appearance panel to display Ground Plane.

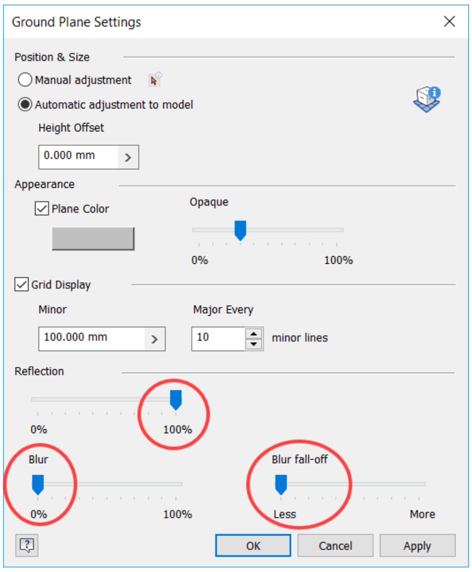




20. Click on **Reflections** icon in Appearance panel to activate reflections > Select **Reflections** settings.



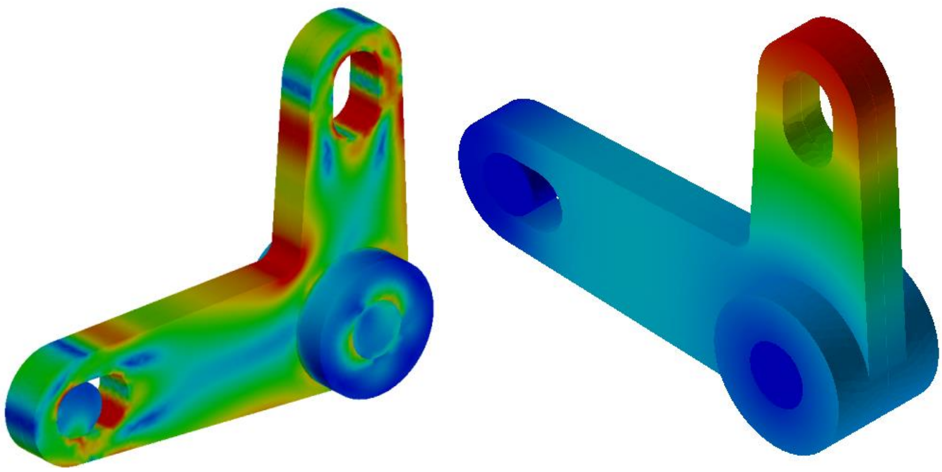
21. Change the Ground Plane Settings as shown below.



Reflection	**100%**
Blur	**0%**
Blur fall-off	**Less**

22. Click **OK**.

23. Click on **Ground Plane** icon again in Appearance panel to hide ground plane > Move the model slightly until you see the reflection as shown below. This gives the impression the full model was analysed as shown below (Stress and Displacement results).



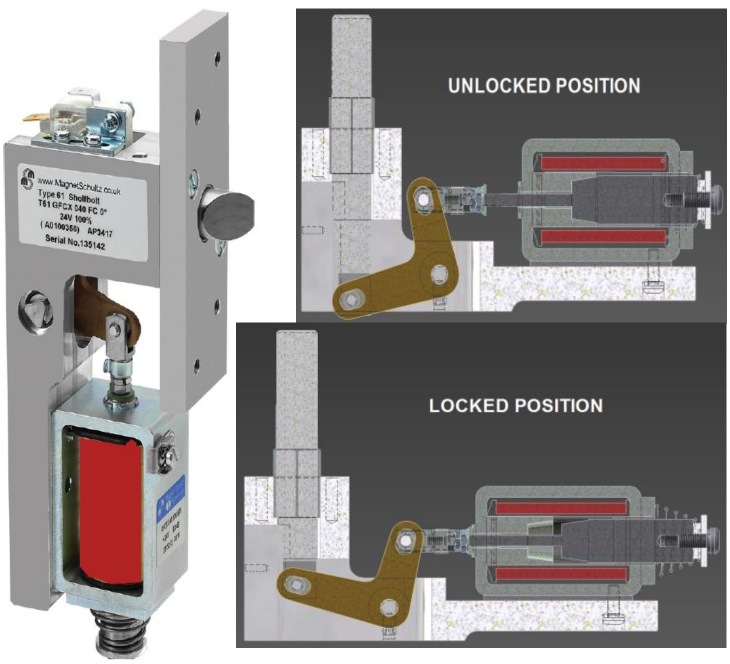
24. **Save** file > **Close** file.

FATIGUE ANALYSIS EXAMPLE

Key features and workflows introduced in this design problem

	Key Features/Workflows
1	**Pin Constraint**
2	**Bearing Load**
3	**Multi-Axis Fatigue Analysis Setup**
4	**Load time based**
5	**Define S-N Curve with supporting information**
6	**Fatigue Results**

Fatigue failures are frequently observed in various components of piston engines. The reasons for such failures can be improper material selection, improper design, misalignment, improper lubrication etc. In this design problem we are going to determine the fatigue life of a connecting rod as shown below.



Design Information

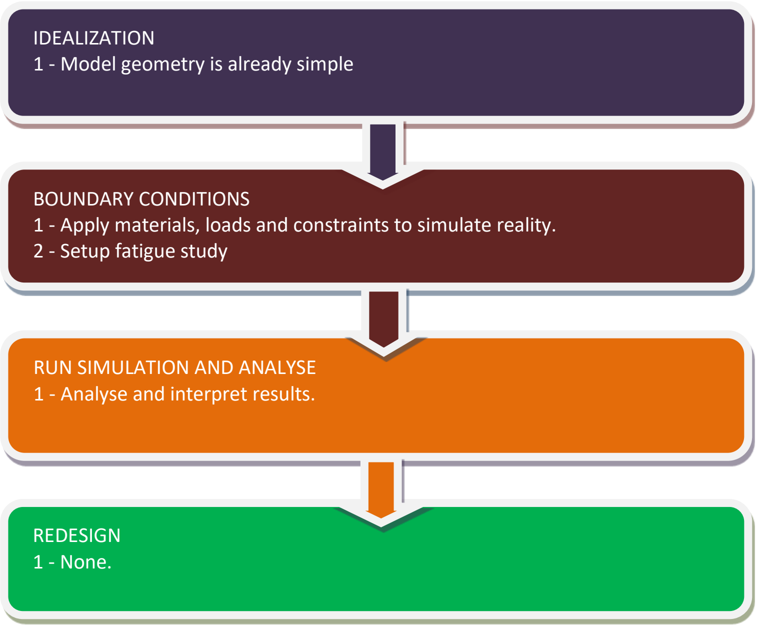
Material of Lever - **Steel**
Density - **7850kg/m³**
Youngs Modulus - **200GPa**
Poisson's Ratio - **0.29**
Yield Limit - **200MPa**
Ultimate Tensile Strength – **340MPa**
Bearing load - **30N** (fully reversible load).

Design Goal

1. To determine fatigue life.

Note. Fatigue life > 1E6

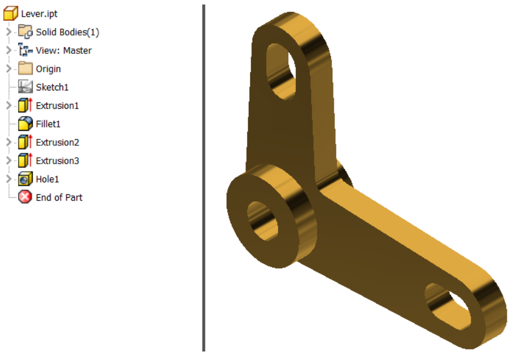
Workflow of Exercise 3



Idealization

The part file is already simple and therefore does not need to go through any further idealization.

1. Open *Lever-Fatigue.ipt.*

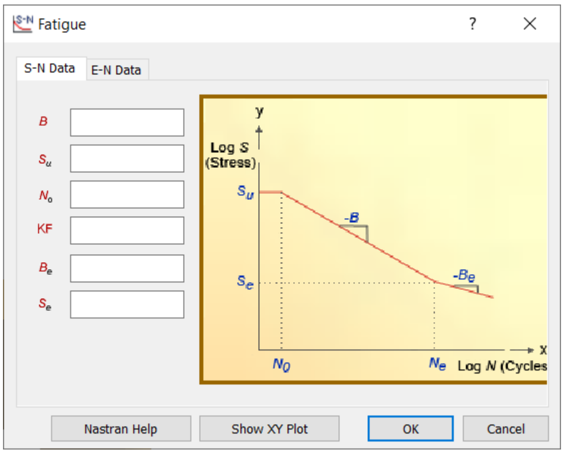


Boundary conditions

2. Select **Environments** tab > Select **Autodesk Inventor Nastran**.

3. **Right Click** Analysis 1 > Select **Edit** > For Type select **Multi-Axial Fatigue**.

4. Click **OK** > Right Click **Steel-DP1** material > Select **Edit** > Select **Fatigue** from Analysis Specific Data.



For ferrous materials, like steel here, you only need to specify the following values.

B – Gradient of the curve in the high cycle region.

S_u – UTS value of material.

N_o – In most cases is always sets to 1000.

S_e – Endurance limit.

To determine B we first need to find the endurance limit. Typically most data is only available for test specimens and is usually denoted as S'_e. For most steel components we can assume the test specimen endurance limit is[1].

$$S'_e = 0.5 S_u \qquad \text{Equation 1}$$

$$S'_e = 0.5 \times 340 = 170$$

To determine actual endurance limit S_e we need to take account of the following factors[2].

$$S_e = k_a k_b k_c k_d k_e k_f S'_e \qquad \text{Equation 2}$$

where

- S_e is the endurance limit for a specific part.
- S'_e is the endurance limit for a test specimen.
- k_a is a surface factor that accounts for the finish (ground, machined, forged, and so on).
- k_b is a size factor that accounts for the size of the part.
- k_c is a loading factor that accounts for different types of loading (bending, axial, torsion).
- k_d is a temperature factor.
- k_e is a reliability factor to account for scatter in the test results from one specimen to another.
- k_f is a miscellaneous factor to account for everything else (residual stress, directional characteristics, corrosion, electrolytic plating, and so on).

In this design problem we are going to assume 1 for all factors except the following.

$Ka = aS_u^b$ Equation 3

Where[3]

Surface Finish	Factor a (MPa)	Exponent b
Ground	1.58	-0.085
Machined or cold-drawn	4.51	-0.265
Hot-rolled	57.7	-0.718
As-forged	272	-0.995

So, using Equation 3 and above values for machined surface, we can calculate Ka.

$Ka = aS_u^b$ => $4.51 \times 340^{-0.265}$ => $4.51 \times 0.2134 = 0.962$

So, using equation 2 we can now determine S_e.

Se = 0.962 x 170 = 163.54MPa.

It also important to note the value of actual S_u at 1000 cycles will be reduced by some factor as illustrated below.

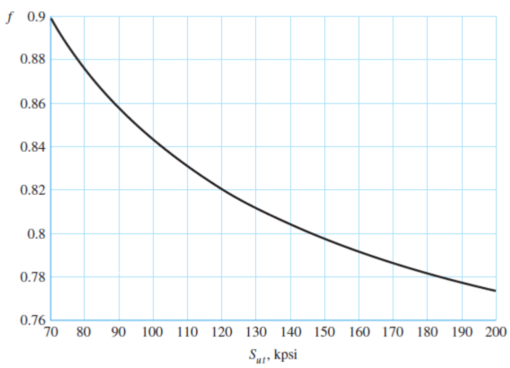
$S_f = f \times S_u$ Equation 5

Where.

f is fatigue strength fraction.

S_f is tensile strength at 1000 cycles.

For components with a Tensile Strength value (UTS) of 482MPa and below we can use 0.9 for *f* as illustrated below[4].



Where

70 kpsi = 482MPa & 200kpsi = 1379MPa

If S_u value is between 70 and 200kpsi then you can take appropriate *f* value from the above.

As our material yield value of 340MPa is below 482MPa we can use a value of 0.9 for fatigue strength fraction to calculate Tensile Strength at 1000 cycles.

$S_f = f \times S_u$ => 0.9 x 340 = 306MPa

References.

1. *(Source obtained from Budynas, Richard G. and Nisbett, J. Keith, Shigley's Mechanical Engineering Design, McGraw-Hill, 9th Edition, 2011,* Section 6-7, page 282)
2. *(Source obtained from Budynas, Richard G. and Nisbett, J. Keith, Shigley's Mechanical Engineering Design, McGraw-Hill, 9th Edition, 2011,* Section 6-9, page 287)
3. *(Source obtained from Budynas, Richard G. and Nisbett, J. Keith, Shigley's Mechanical Engineering Design, McGraw-Hill, 9th Edition, 2011,* Table 6-2, page 288)
4. *(Source obtained from Budynas, Richard G. and Nisbett, J. Keith, Shigley's Mechanical Engineering Design, McGraw-Hill, 9th Edition, 2011,* Figure 6-18, page 285)

We now have the following values.
S_u – 306MPa

N_o – 1000.

S_e – 163.54MPa.

We now have all the information required to calculate B from the following equation.
The slope of the S-N curve is

$$B = \frac{\log(S_u) - \log(S_e)}{\log(N_e) - \log(N_0)}$$

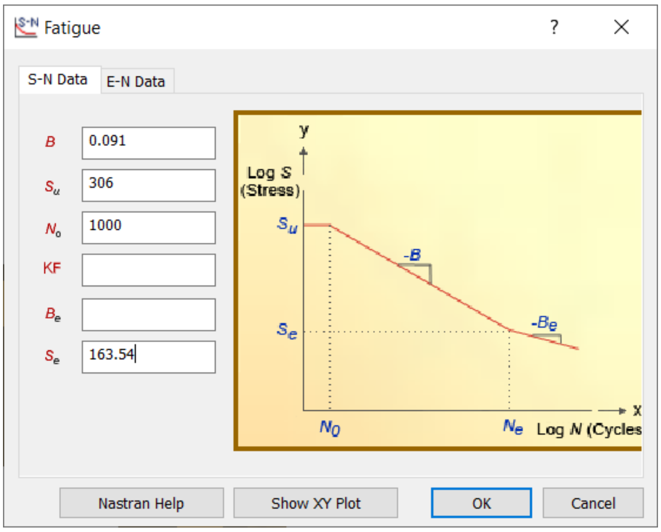
Where Ne is usually 1 million cycles.

$$B = \frac{\log(306) - \log(163.54)}{\log(1E6) - \log(1E3)}$$

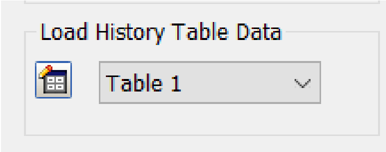
$$B = \frac{2.486 - 2.214}{6 - 3}$$

$$B = \frac{0.272}{3} = 0.091$$

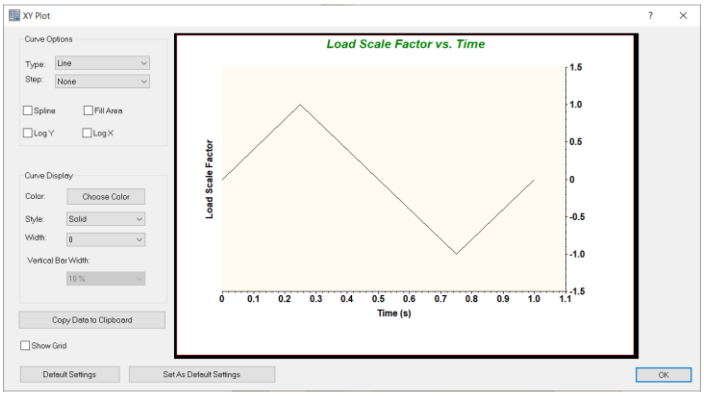
5. Specify **0.091** for B > Specify **306** for S_u > Specify **1000** for N_o > Specify **163.54** for S_e.



6. Click **OK** twice > Double Click **Pin-Load >** Select **Table 1** from the Load History Table Data.



7. Select **Edit Table Data**.> Select **Show XY Plot.**

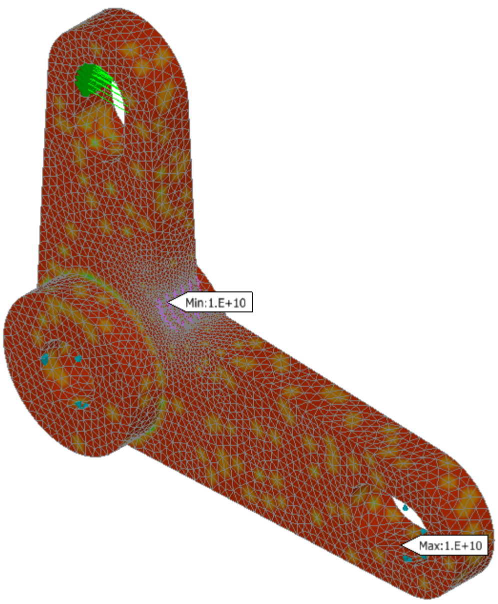


8. Click **OK** three times.

Run simulation and analyse

9. Select **Run** > Click **OK** once run is complete.

10. Double Click **Life Contour** in the browser.



Minimum Life is 1e10.

Now try running Fatigue again by changing the load from -30N to -500N. **What is the Fatigue life now?**

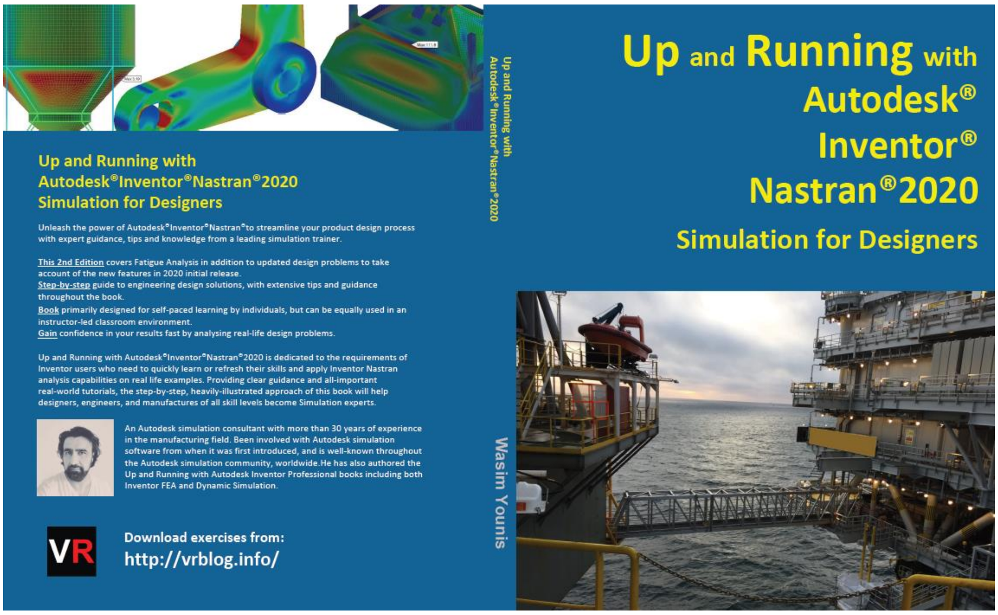
11. **Close** File.

ADDITIONAL RESOURCES

The exercises, including the step by step instructions and guidance, used in this handout have been taken from my Up and Running with Autodesk® Inventor® Nastran® 2020 book. The book covers many more examples you can go through as outlined below.

- Chapter 1 - Introduction to Inventor Nastran Environment
- Chapter 2 - Part Analysis of a Lever Arm
- Chapter 3 - Part Analysis of a Propshaft Yoke
- Chapter 4 - Assembly Analysis of a Lever
- Chapter 5 - Assembly Analysis of a Bolted Bridge Structure
- Chapter 6 - Shell Analysis of a Seed Hopper
- Chapter 7 - Shell Analysis of a Base Frame
- Chapter 8 - Beam Analysis of a Maintenance Platform
- Chapter 9 - Beam Analysis of a Gangway
- Chapter 10 - Shell/Beam Analysis Water Tank
- Chapter 11 - Fatigue Overview
- Chapter 12 - Fatigue Analysis of Rotating Shaft
- Chapter 13 - Fatigue Analysis of Connecting Rod

Up and Running with Autodesk® Inventor® Nastran® 2020 is dedicated to the requirements of Inventor users who need to quickly learn or refresh their skills and apply Inventor Nastran analysis capabilities on real life examples. Providing guidance and all-important real-world tutorials, the step-by-step, heavily-illustrated approach of this book will help designers and engineers of all skill levels become Simulation experts.



The book is available worldwide through Amazon sites. Below are further links to get extra help and guidance to get you up and running with Inventor Nastran.

Inventor Nastran Forum -
https://forums.autodesk.com/t5/nastran-in-cad-forum/bd-p/75
Inventor Nastran Online Tutorials –
http://help.autodesk.com/view/NINCAD/2019/ENU/?guid=GUID-DB7160BE-0C72-47B9-B5EF-FC4925B455CE