AUTODESK UNIVERSITY 2015

IM10370

# What do my answers mean? All about the results in FEA

Jim Byrne
Autodesk

## Handout Objectives

- Stress and factor of safety plots
- Options for contour plots
- Common assumptions to look out for in results

## Description

Post processing results from finite element analysis (FEA) solutions is essential to understanding whether your design passes or fails based on your engineering criteria. There are many options available for checking the design integrity, such as different types of stress plots. There are also several contour options such as nodal, elemental, max, and average. During this course we'll explain the importance of all these options and what to look out for to make sure the results are accurate.

## Your AU Experts

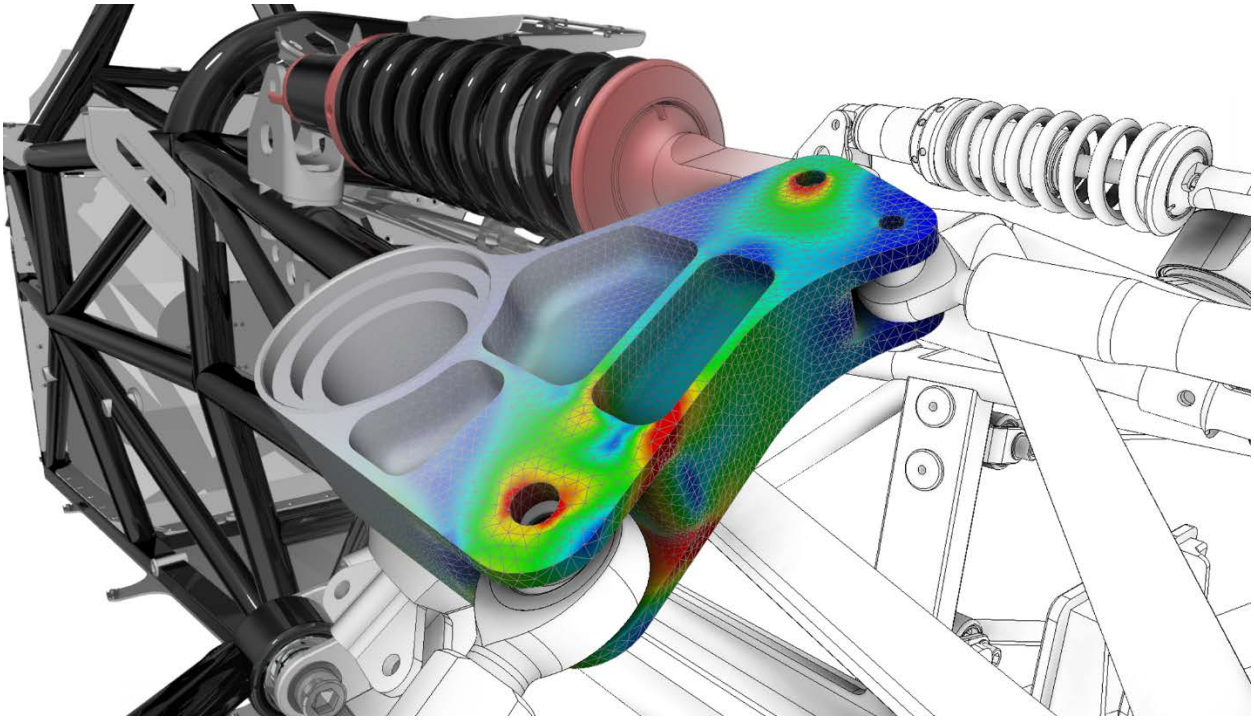
*Jim Byrne joined Autodesk in 2013 as a Manufacturing Technical Marketing Specialist. He is responsible for producing marketing content for Simulation products for FEA and CFD as well as content creation for Factory Design Suite.*

*Prior to joining Autodesk, Jim worked for a local reseller for 14 years selling and supporting CAD, Simulation, and data management solutions for several companies in Michigan, Indiana, and Ohio. Jim also has three years of experience in the industry as a machine designer.*

## Introduction

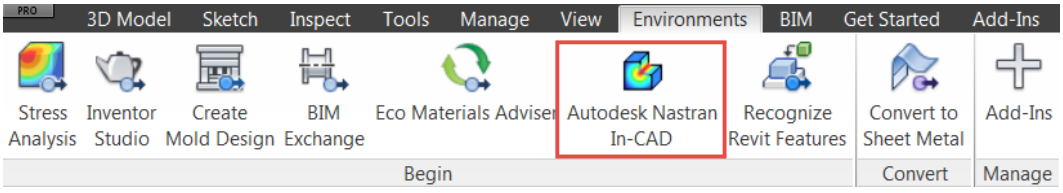
In the following case study we are conducting an analysis on the rocker for the front suspension of the BAC Mono high performance vehicle. This is just one of several example files we will be using in the course.

The rocker is taking a large load from the front wheel. Are the stresses low enough to avoid the material from yielding? Will the part last? Should we make changes to strengthen the component? Or should we remove material if it is overbuilt?

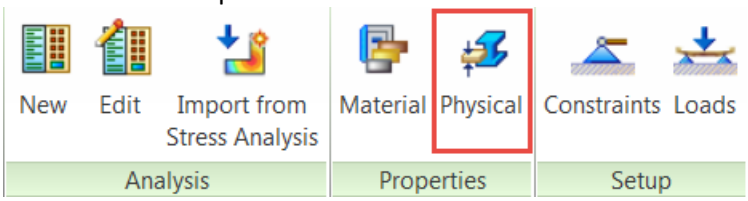


## Setup the Study

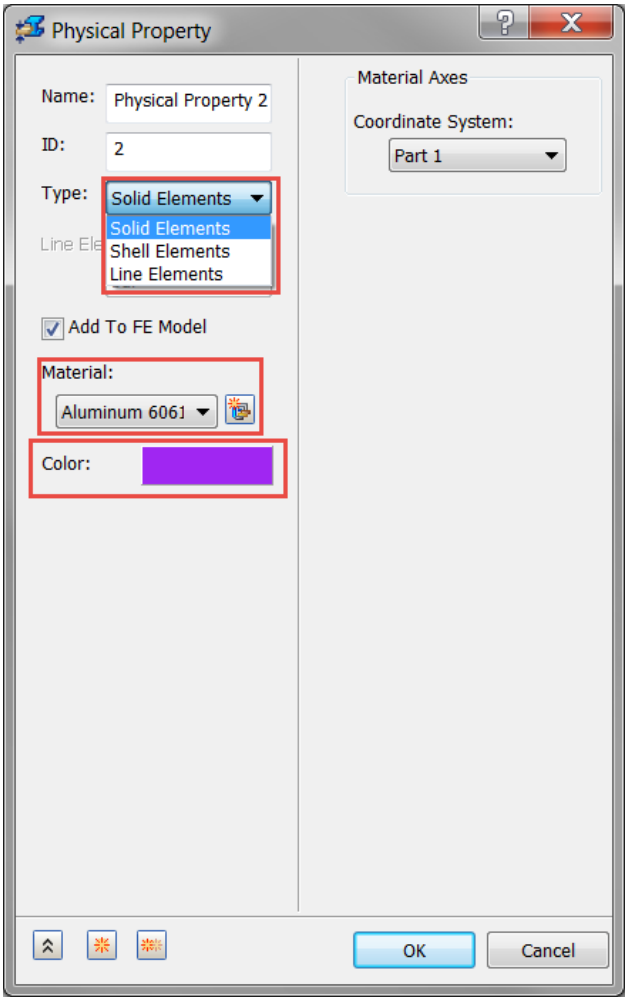
1. Open “Front Rocker (start).ipt”
2. Turn on Autodesk Nastran In-CAD in the Environment ribbon



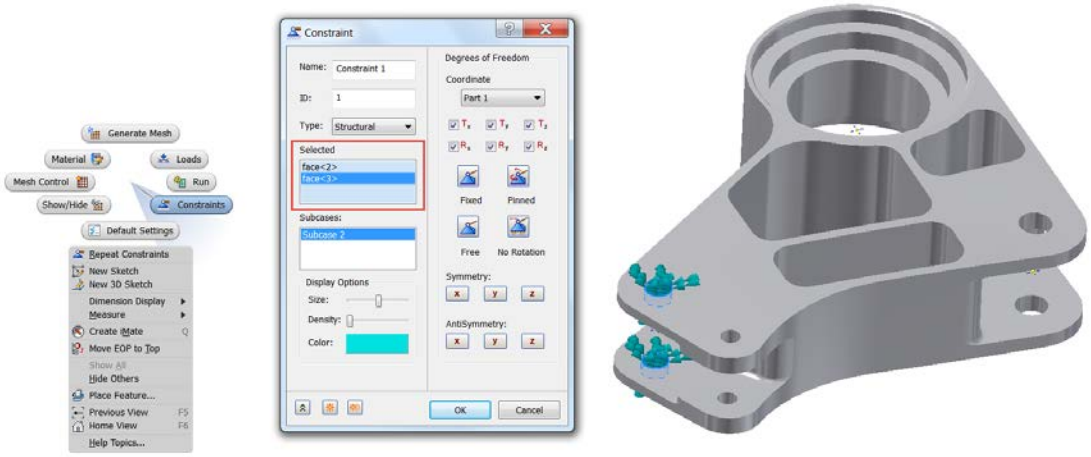
3. Add a physical property to the front rocker. This will determine the element type and material used for the component.



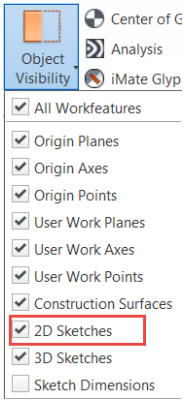

4. Accept the defaults in the physical property dialog or feel free to change the material and color.



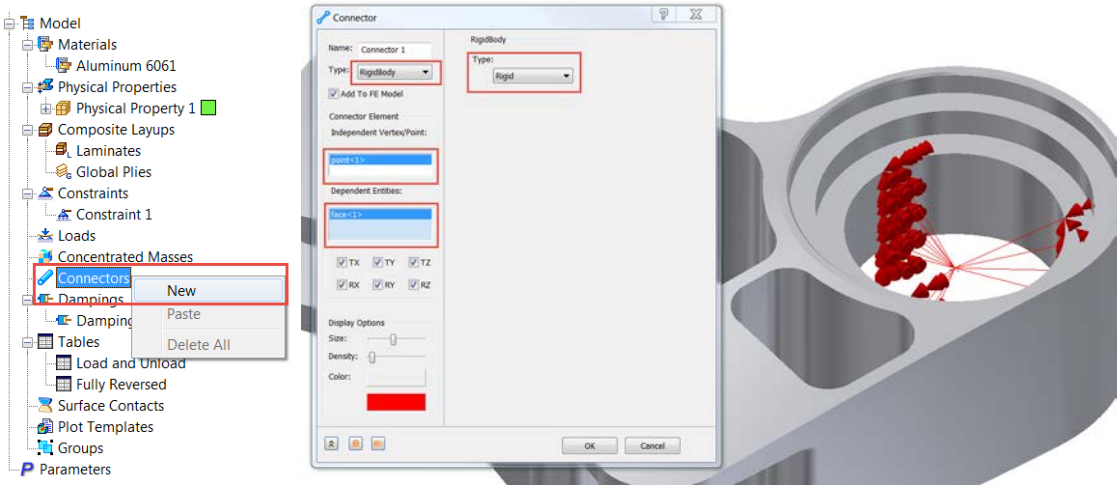
5. Add fixed constraints to the two holes that attach to the damper in the front suspension.



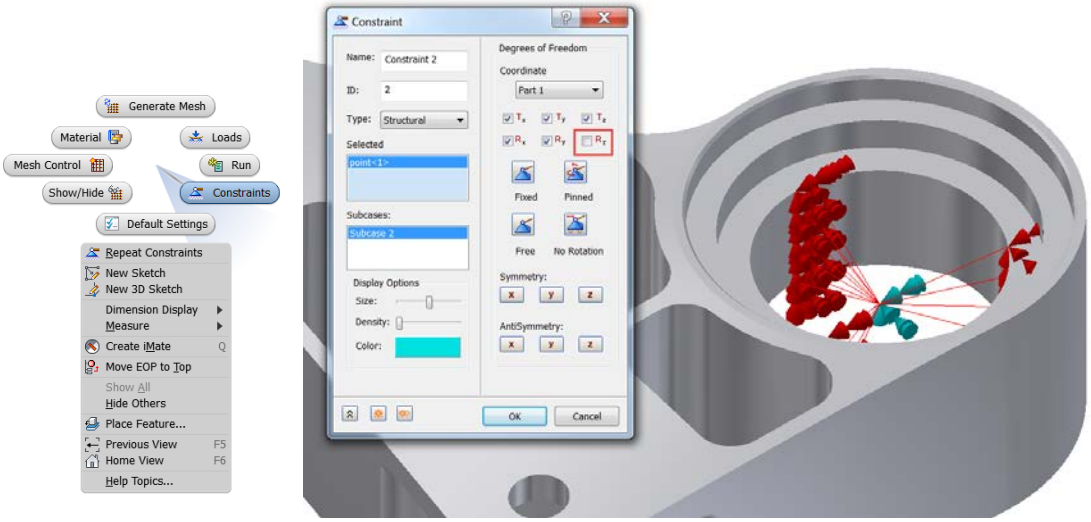

6. Turn on 2D sketches and turn off sketch dimensions for clarity.



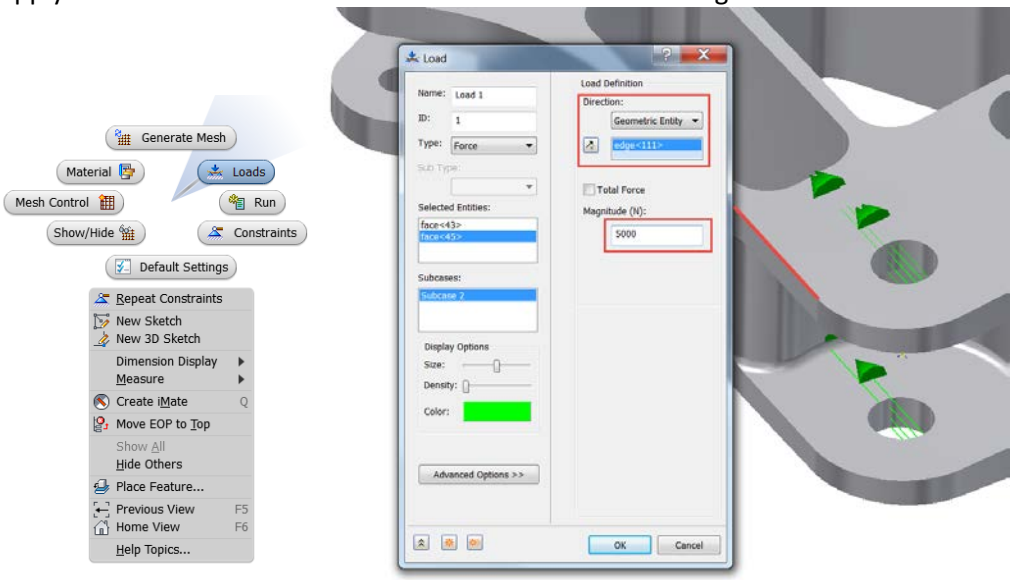

7. Add Rigid Body connectors to the main bore of the front rocker. This applies line elements or a rigid connection between the center point and the cylindrical face.



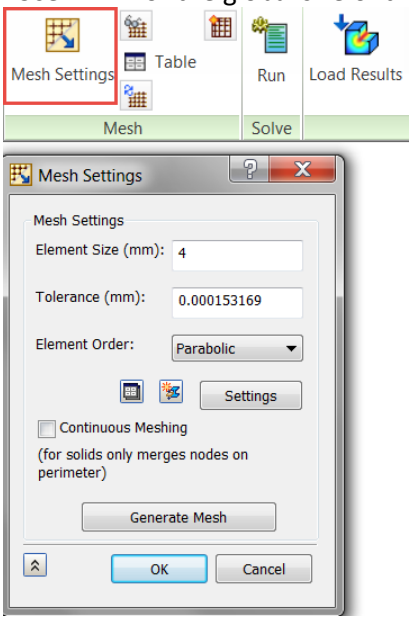
8. We can now define the behavior of the main bore in many ways by constraining any of the six degrees of freedom at the center. Add a constraint to the center point of the main bore. Leave the Z Rotation unchecked. Only the axial direction has the freedom to rotate.



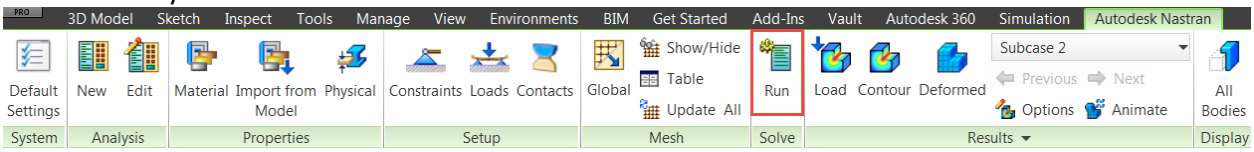

9. Apply a force of 5000 Newtons to the two holes. Use the edge as a direction.



10. Use 4mm for the global size of the mesh.

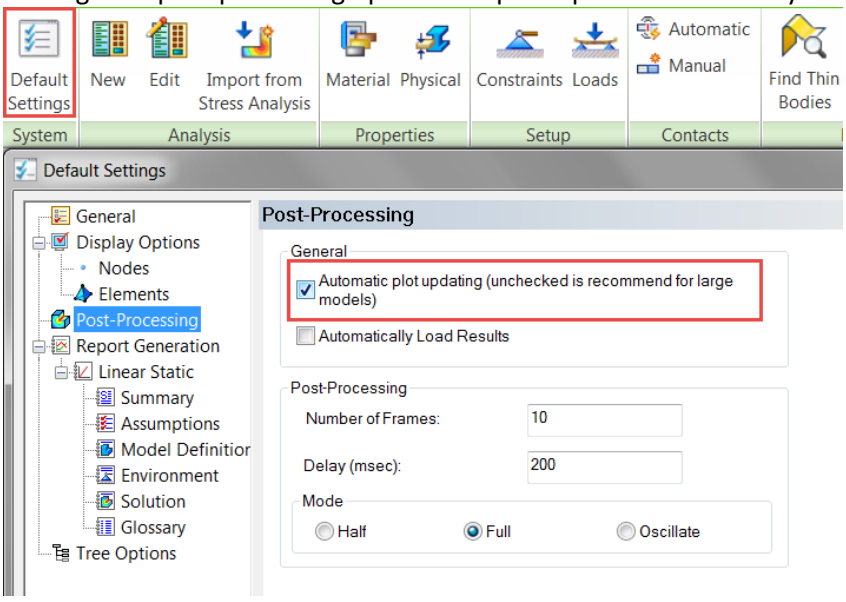



11. Run the study

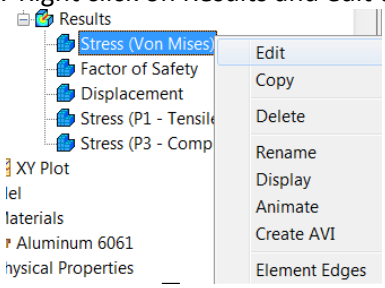



## Contour Plot Options

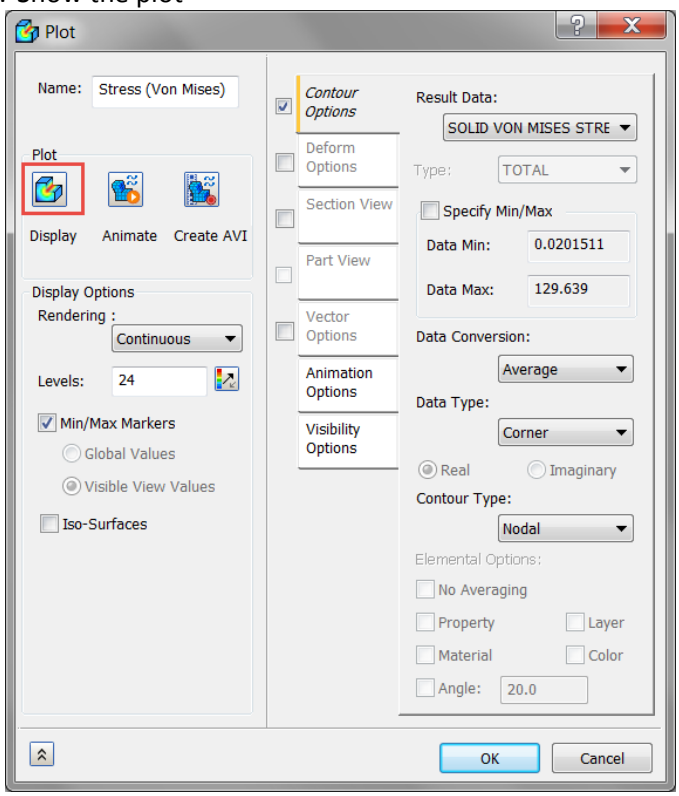
12. Change the post-processing options to update plots automatically after changing plot settings.



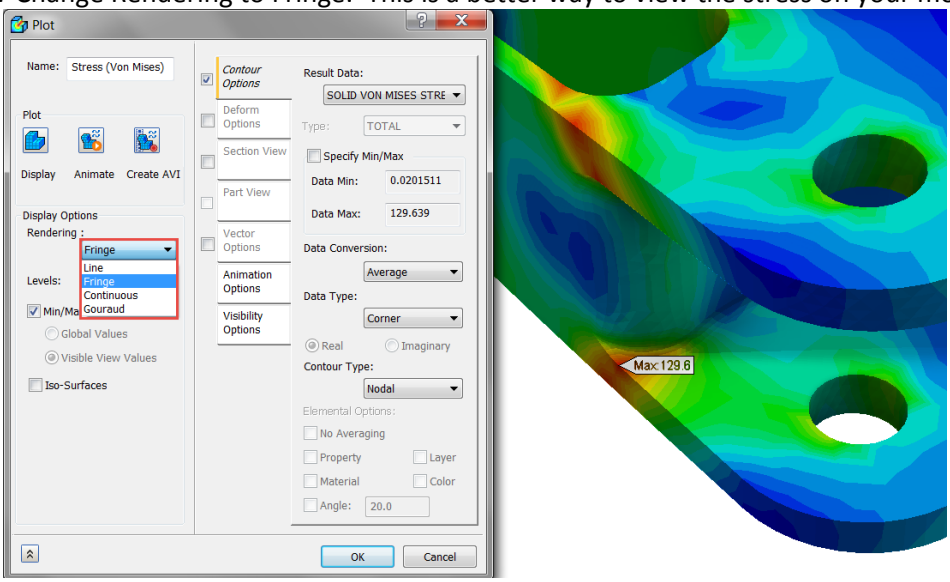
13. Right click on Results and edit the von Mises stress plot



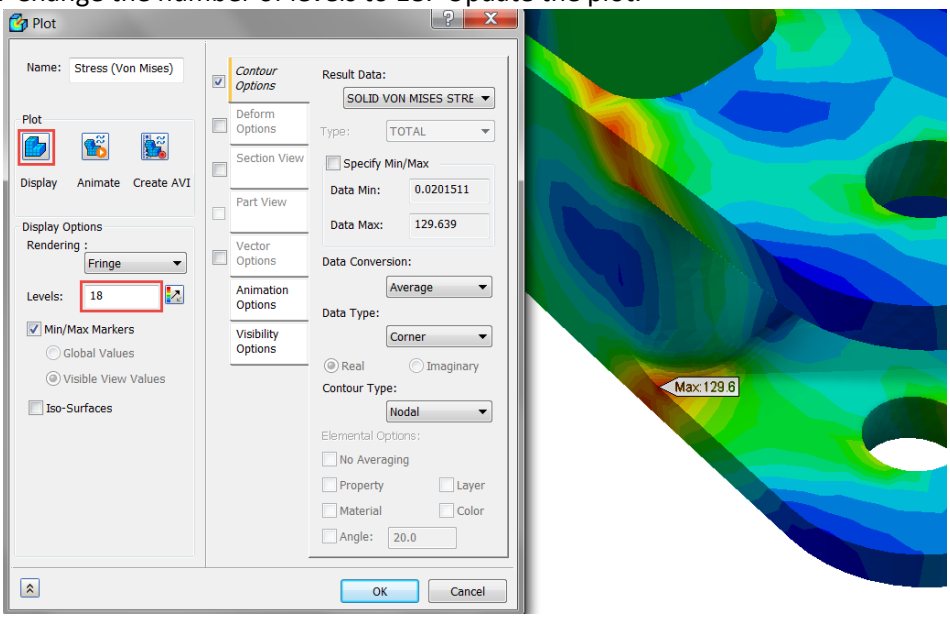

14. Show the plot



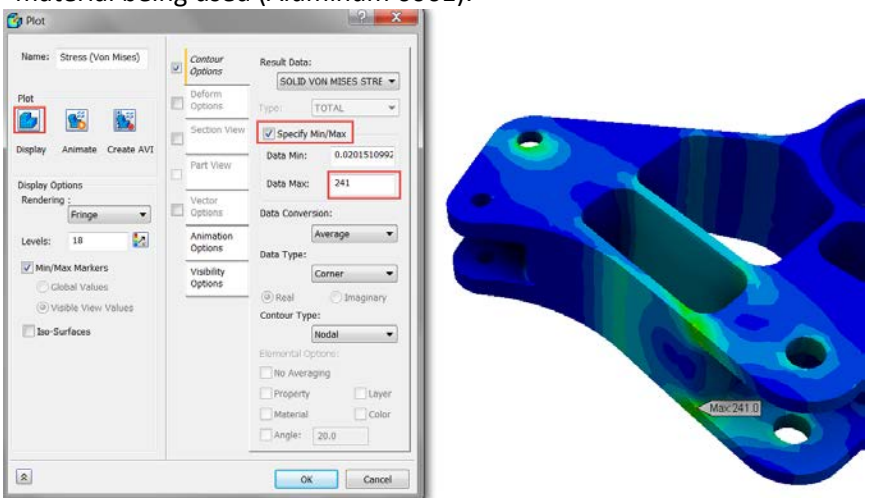
15. Change Rendering to Fringe. This is a better way to view the stress on your model.



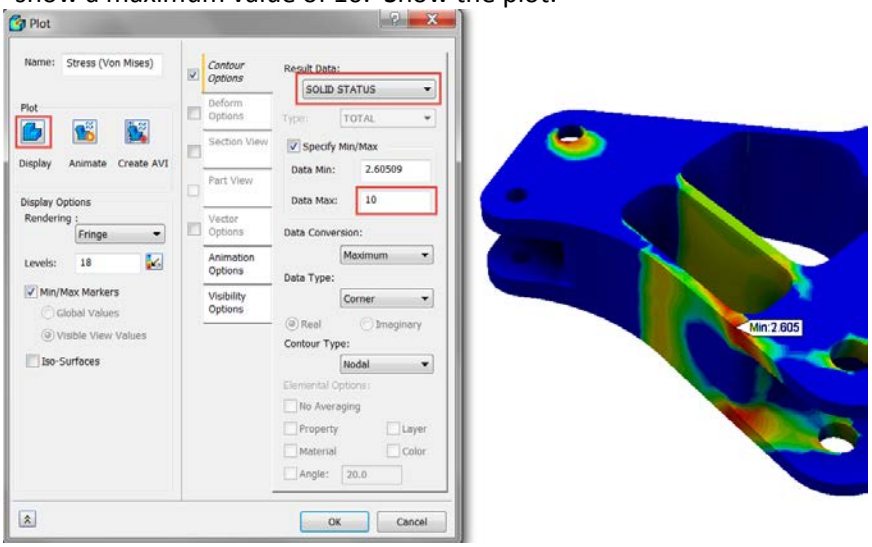

16. Change the number of levels to 18. Update the plot.



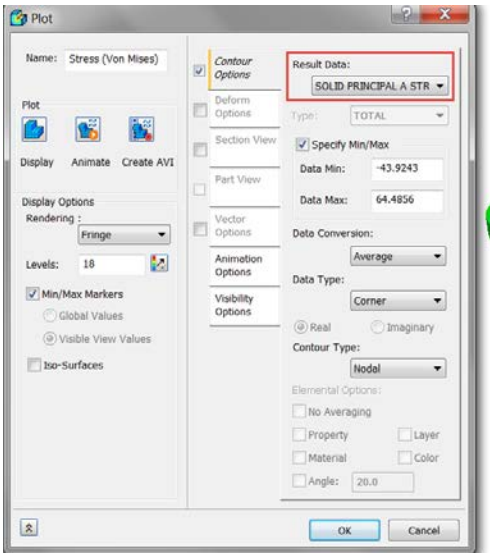
17. The areas that are red do not indicate failure. It is simply the highest calculated stress on the model. Change the legend to 241mPa for the Data Max. This is the yield strength of the material being used (Aluminum 6061).

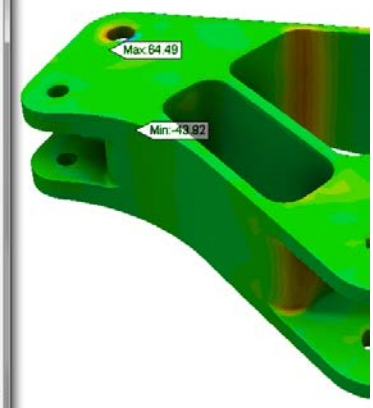



18. Check the factor of safety. Change the result data to “Solid Status”. Change the legend to show a maximum value of 10. Show the plot.

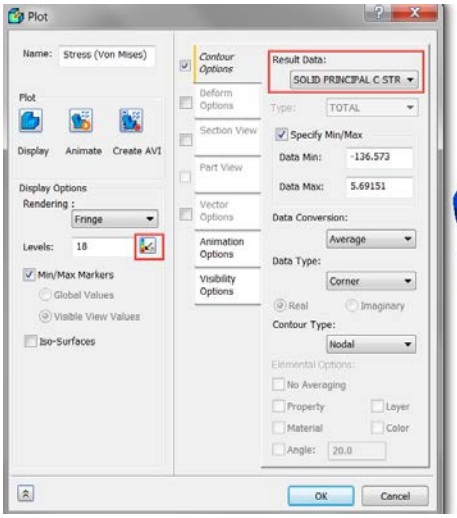


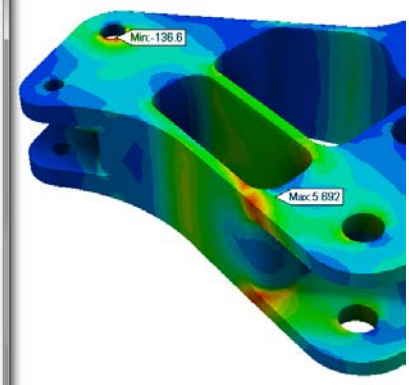
19. Change the result data to Solid Principal A Stress to view the maximum tensile stresses in the model.



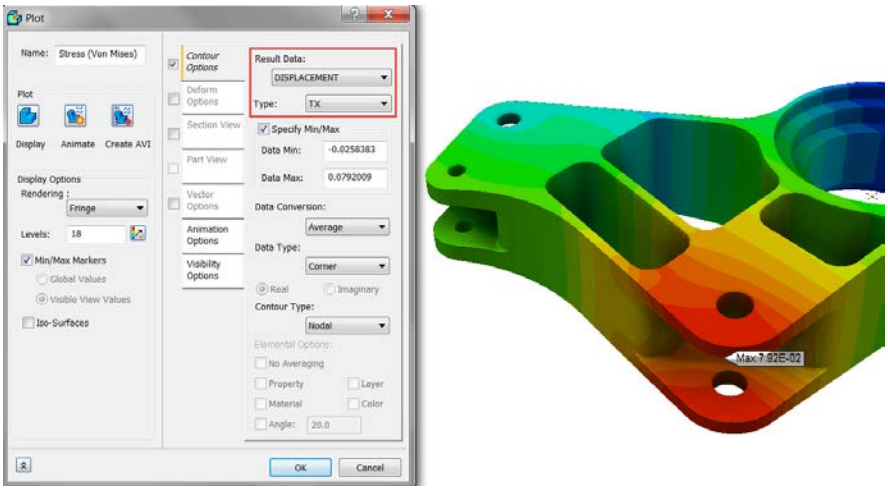

20. Change the result data to Solid Principal C Stress to view the maximum compressive stresses in the model. In this plot we are looking for the lowest value on the legend which will be a negative number. Optional: Flip the colors in the legend to show the negative compressive stresses in red.



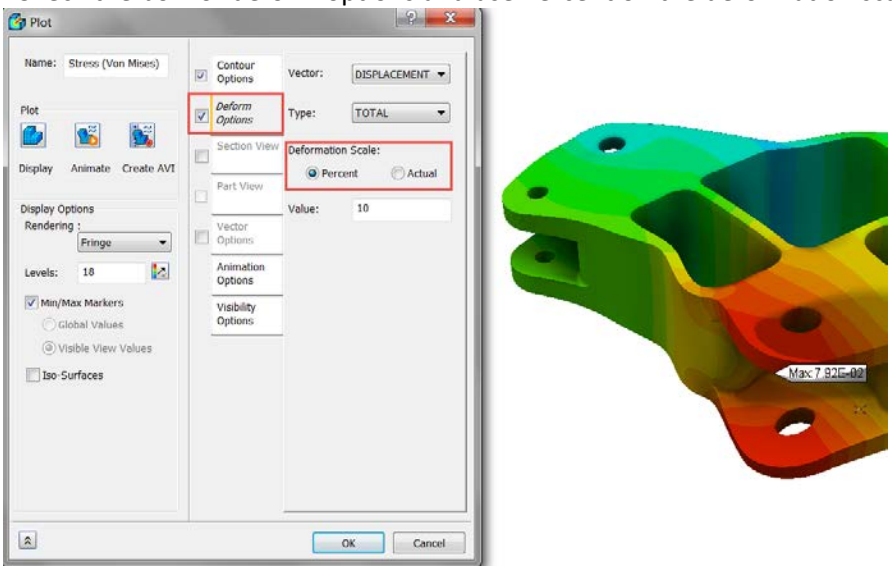



## Displacement and Animation

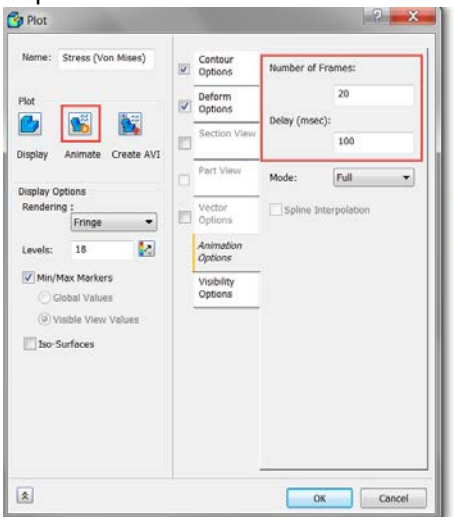
21. Change the result data to Displacement. Also change the Type to TX to view the results in just the X direction.

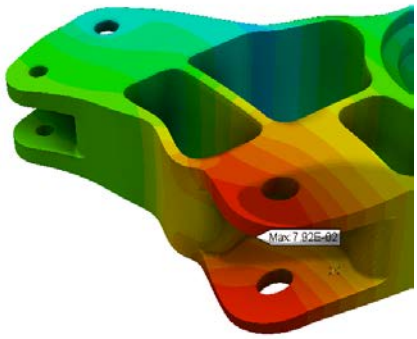



22. Check the box for deform options and use Percent on the deformation scale.

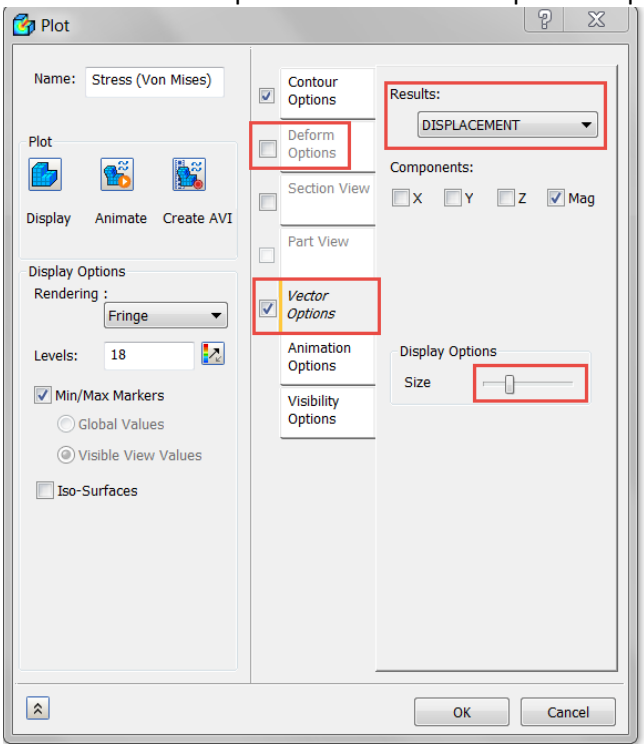


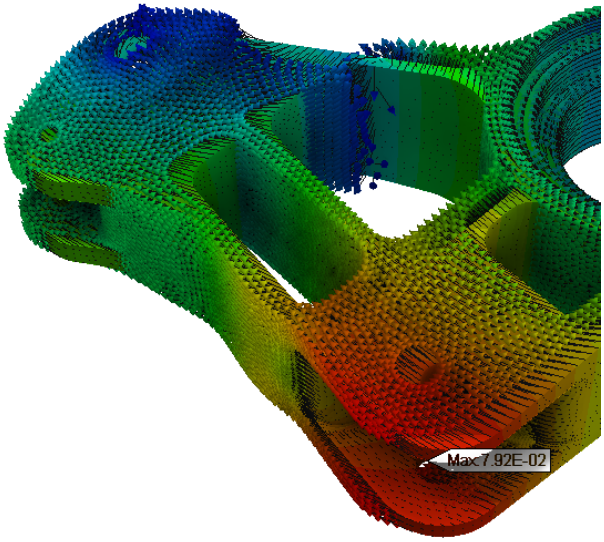
23. Change the number of frames to 20 and delay to 100 milliseconds. Click on “Animation Options”.




24. Turn off Deform Options. Check Vector Options. Optional: Change the size of the arrows.

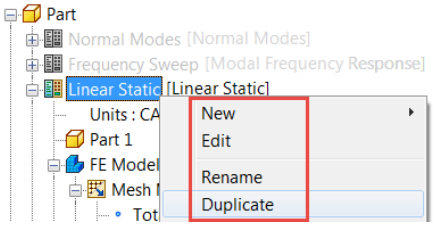


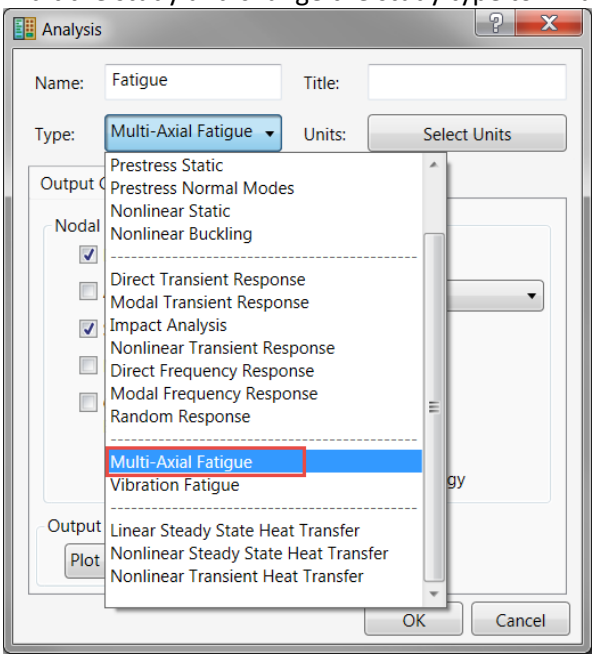
*The linear static study shows that the front rocker will pass with a factor of safety of two. Basically that means it can withstand a load of up to 10,000 Newtons. Should we stop there? Or should we consider other factors such as the number of loads that are being applied? It is possible that after applying the load thousands of times that cracks will begin to form on the part. Let's run a fatigue study to find out how long the front rocker will last.*

## Fatigue

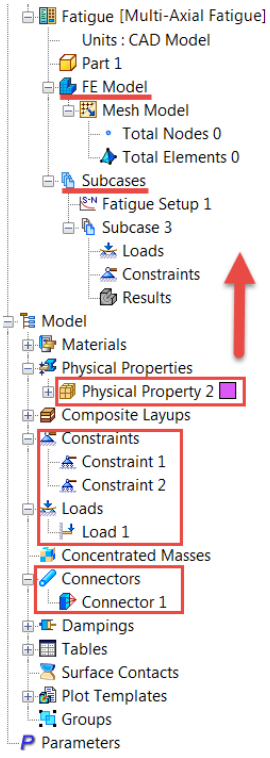
1. Create a new study and rename it to "Fatigue". Or drag and drop the "Linear Static" study to the top of the browser to create a copy of the existing study along with copies of the physical property and all boundary conditions.



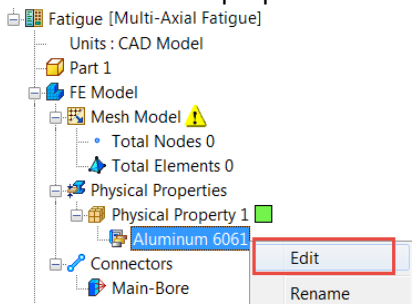

2. Edit the study and change the study type to "Multi-Axial Fatigue".



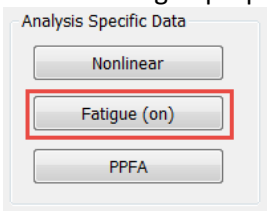
3. *Skip this step if you duplicated the "Linear Static" study.*
   If you created a new study then drag the physical property and boundary conditions from the model tree to the new "Fatigue" study.



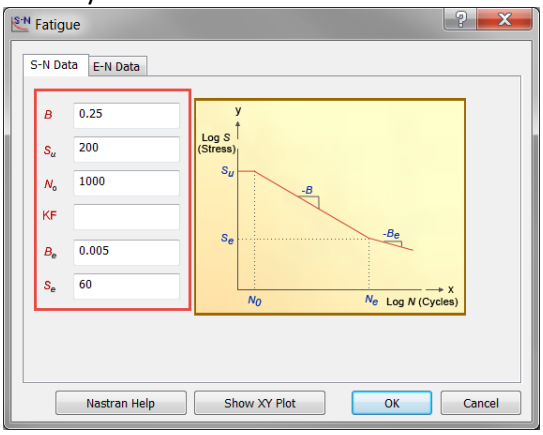

4. Edit the material properties.



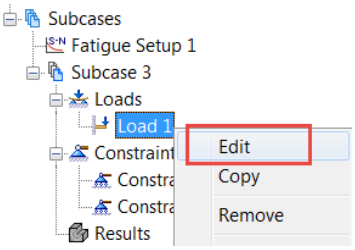

5. Edit the fatigue properties.



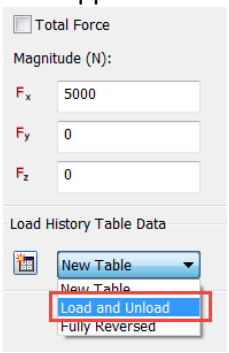
6. The fatigue curve is already defined for you. Fatigue studies need the slope of the fatigue curve, a point on the curve, endurance limit, and slope after the endurance limit which is usually close to zero.



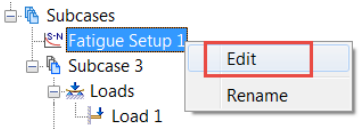

7. Edit the load.



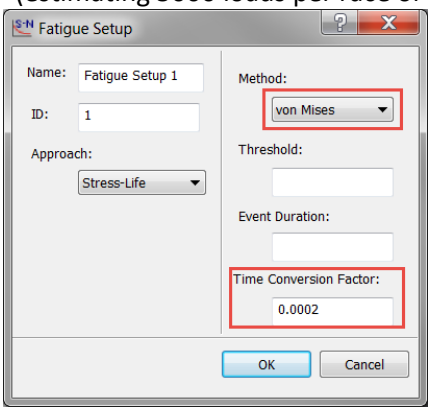

8. Change the load condition to load and unload. This is a predefined table that applies the full load and then releases it. Choose the other predefined table if the same load is also applied in the opposite direction.



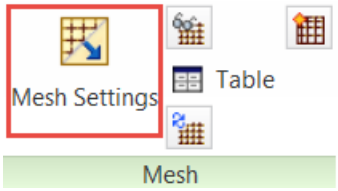

9. Edit the fatigue setup.



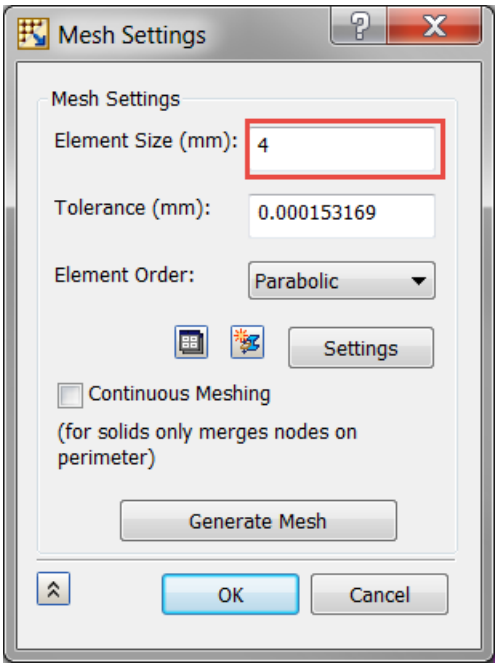
10. Change the stress results used in the fatigue study to vonMises. Also change the time conversion factor to 0.002. This converts the number of loads to the number of races (estimating 5000 loads per race or the reciprocal 1/5000).




11. Select "Mesh Settings"

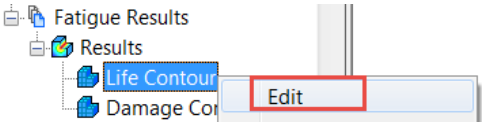



12. Use an element size of 4mm.

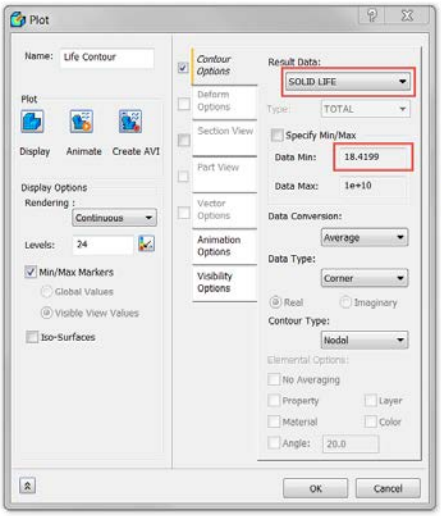


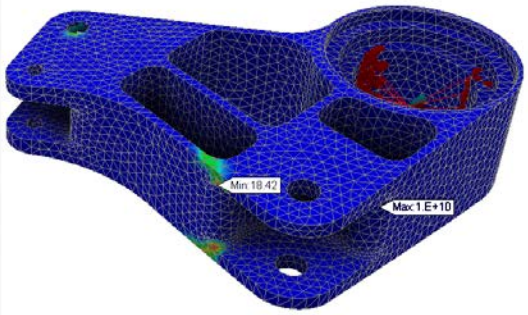

13. Run the study.

14. Edit the “Life Contour” plot

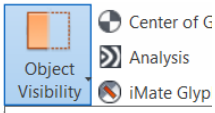


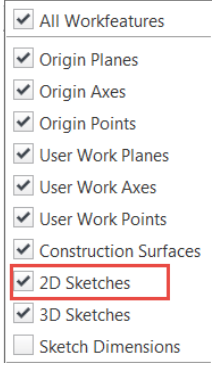

15. Change the plot type to “Solid Life”. Optionally reverse the legend colors. Notice the rocker will only work for 18 races at 5,000 loads per event.



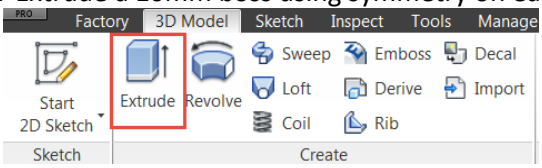



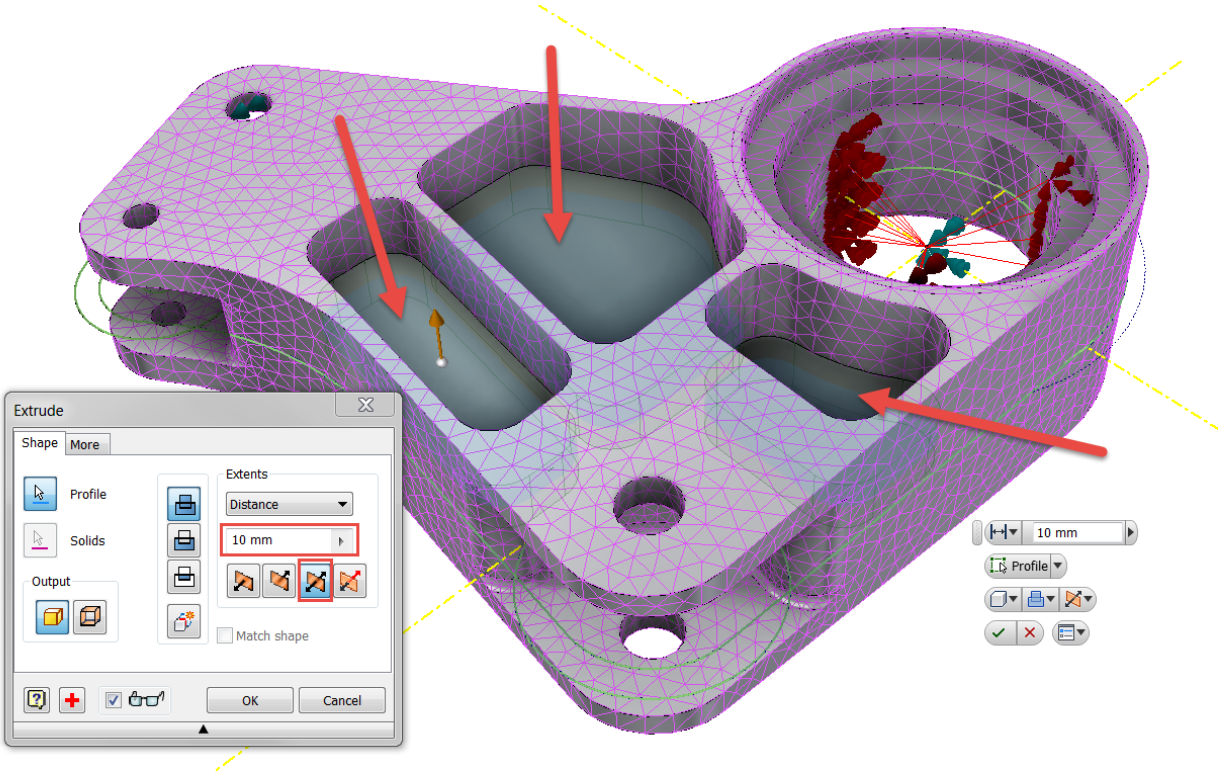

16. Turn on 2D Sketches in the Object Visibility pulldown in the View tab.



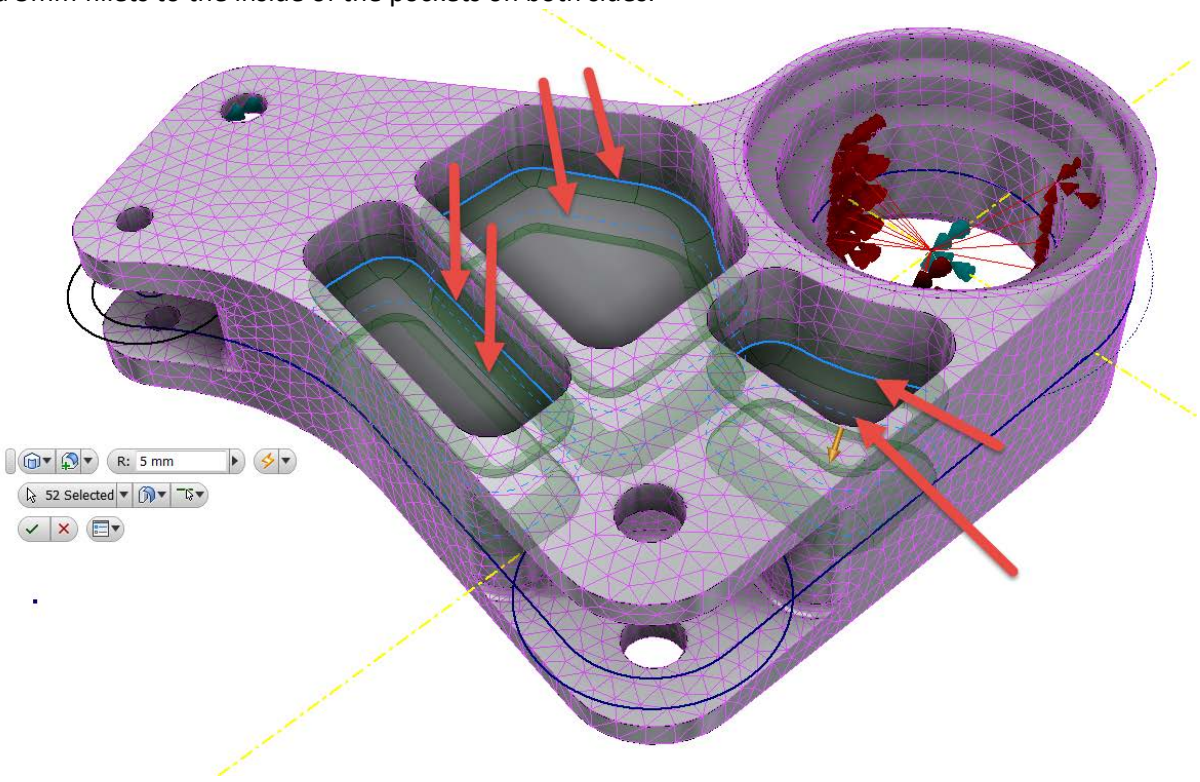




17. Extrude a 10mm boss using symmetry on each of the three pockets.

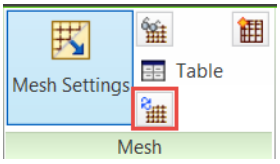




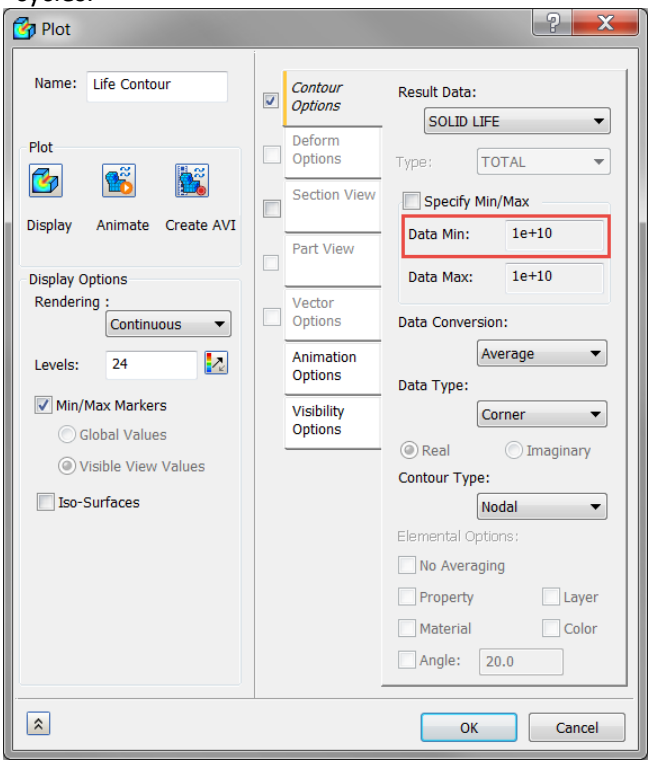

18. Add 5mm fillets to the inside of the pockets on both sides.



19. Generate a new mesh for the updated geometry.



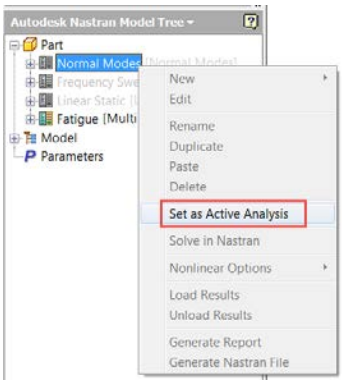

20. Run the study.
21. Edit the “Life Contour” plot.
22. Notice that the minimum life of the rocker is shown as 1e+10. This is a good indication we are within the endurance limit of the material. The part will last for an infinite number of load cycles.



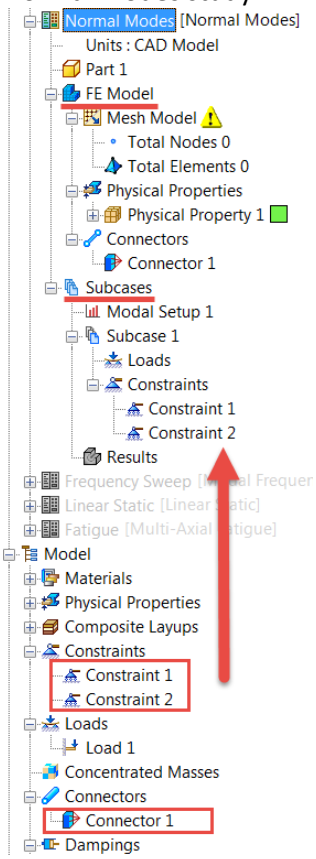
## Normal Modes

*Now we know that we have a model that will not only work under a single load. We know it will last even under thousands of load cycles over the life of service. That being said, the two studies that we ran are using static boundary conditions. In reality the vehicle will be moving. We should consider the effects of vibration. Will the front rocker shake too much? Let's find out with Normal Modes and Transient Response studies.*

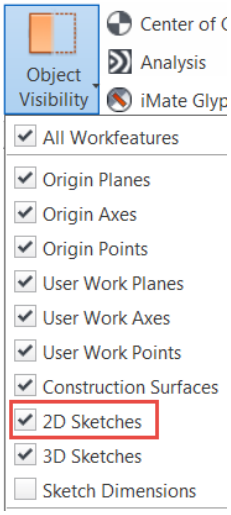
1. Activate "Normal Modes"



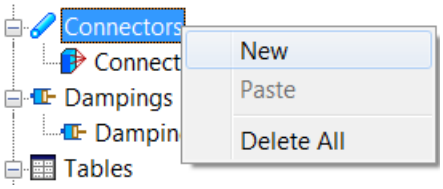

2. Copy the Constraints and Rigid Body Connector that was used in the Linear Static study to the Normal Modes study.



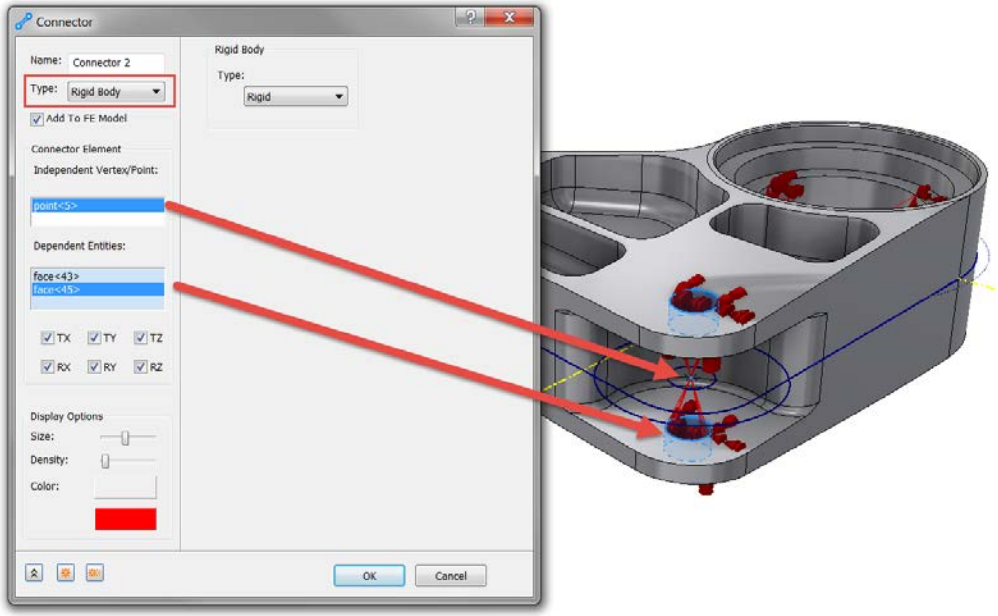
3. Turn on 2D Sketches



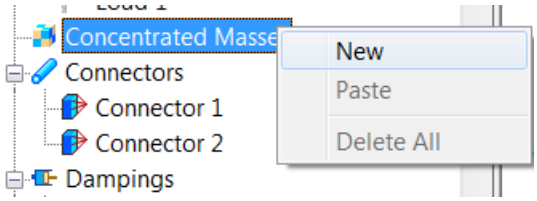

4. Add a new Connector



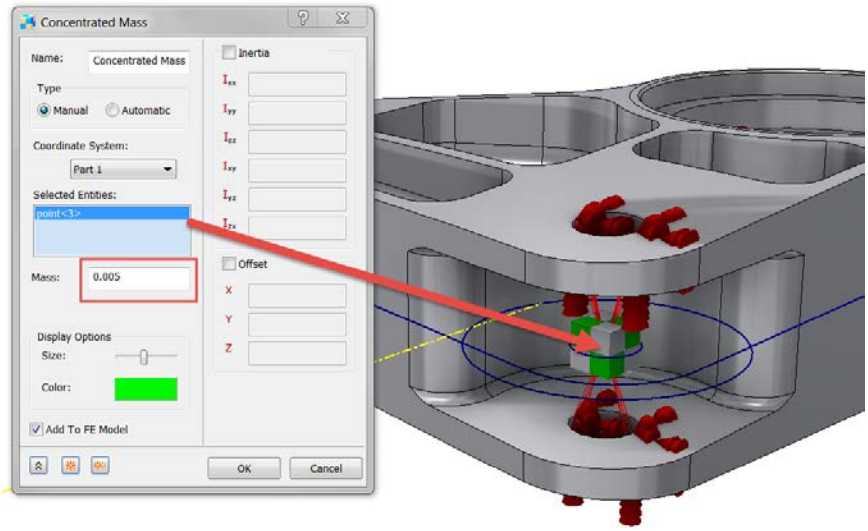

5. Apply a Rigid Body connector between the point on the 2D sketch to the two faces of the hole.



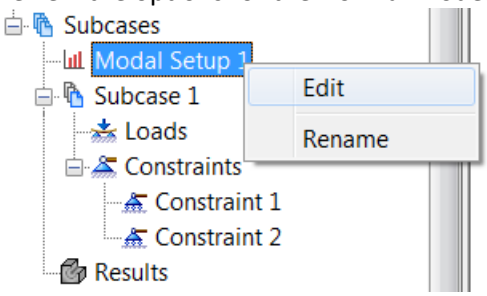
6. Add a new Concentrated Mass.



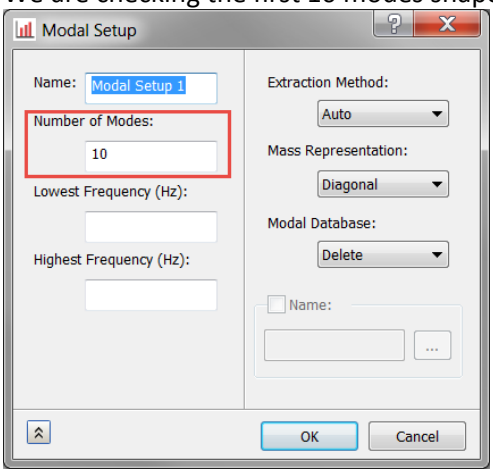

7. Apply the concentrated mass to the 2D sketch point.



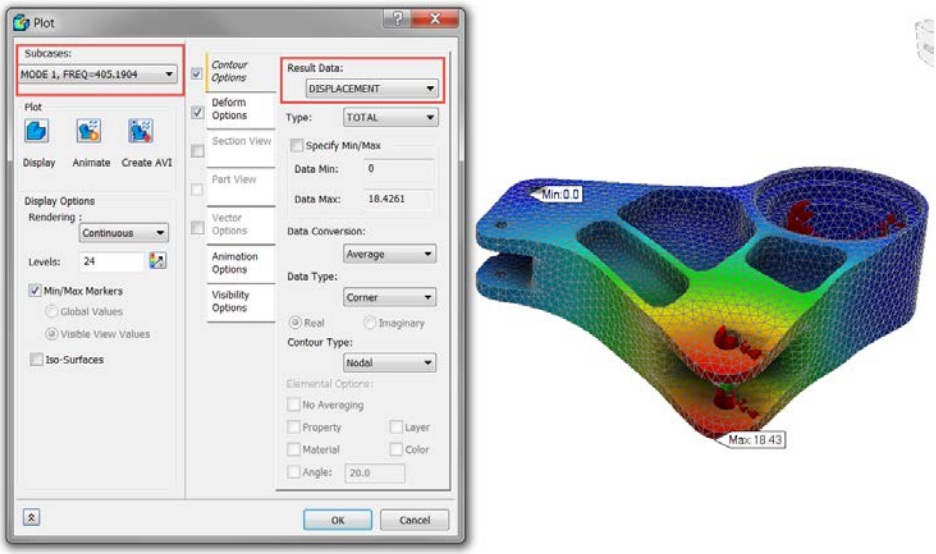

8. Review the options for the Normal Modes study.



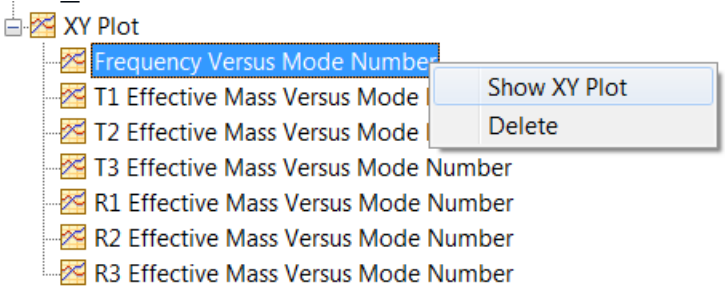
9. We are checking the first 10 modes shapes and frequency levels for the front rocker.

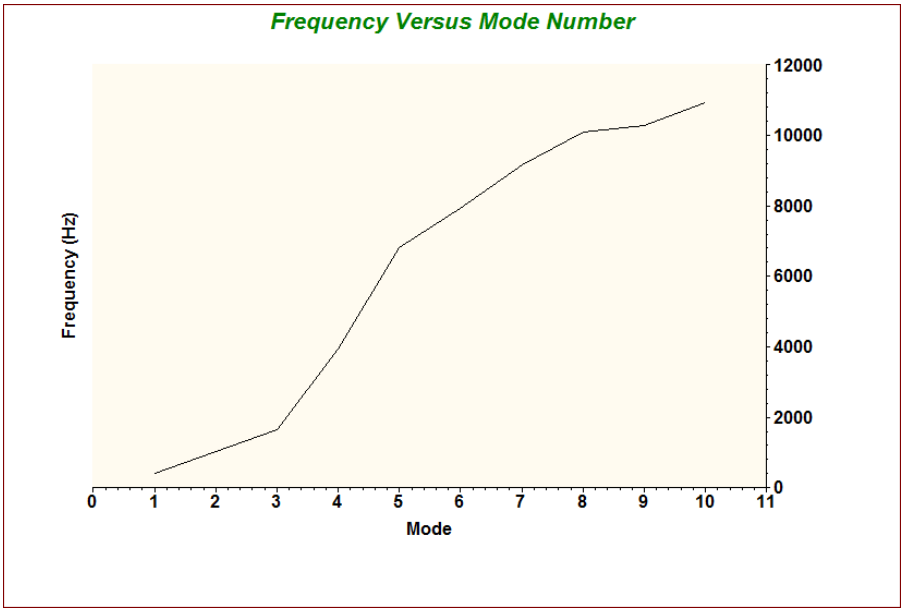



10. Mesh and Run
11. Right click on results and check the behavior of the first mode shape. The frequency is 405 Hz. The displacement value is not the actual distance. We are just using the displacement plot to show where the maximum displacement is occurring.




12. View the frequency for each of the mode shapes in a graph.

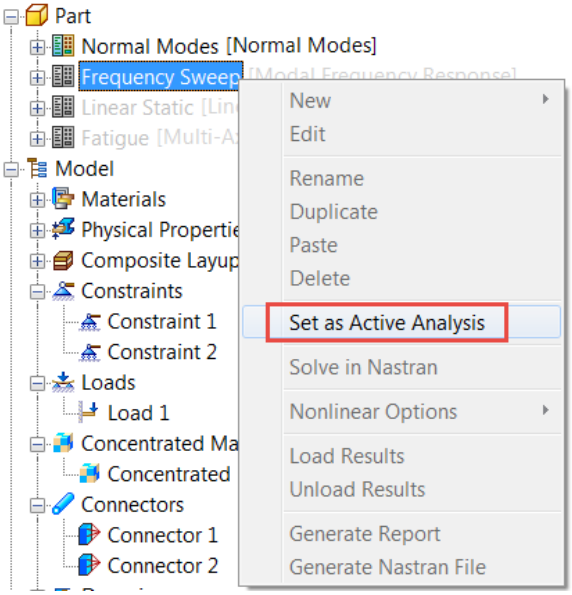




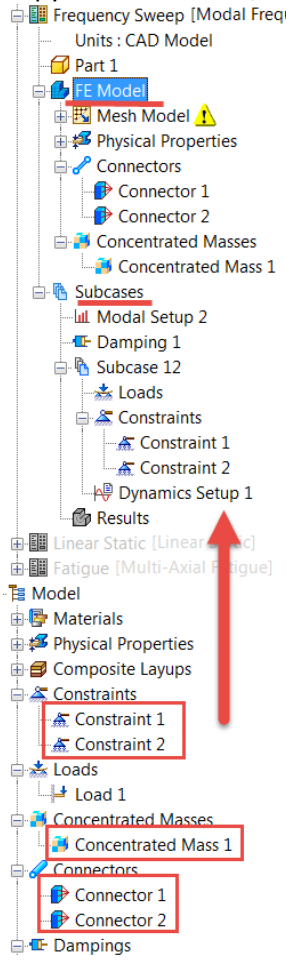

## Frequency Response Study

Which of the mode shapes should we be concerned with? Let's run a Frequency Response study to gain more insight into the front rocker with the addition of an applied acceleration that is normal to the road. This study will provide us the maximum stress and displacement results for a specified range of frequencies between 0 and 5000 Hz. Also known as a frequency sweep.

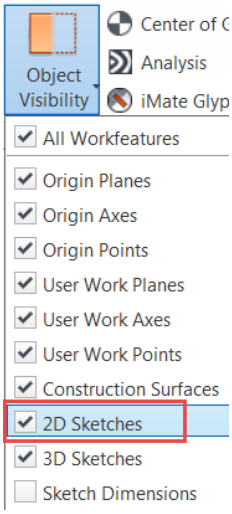
1. Activate the "Frequency Sweep" study.



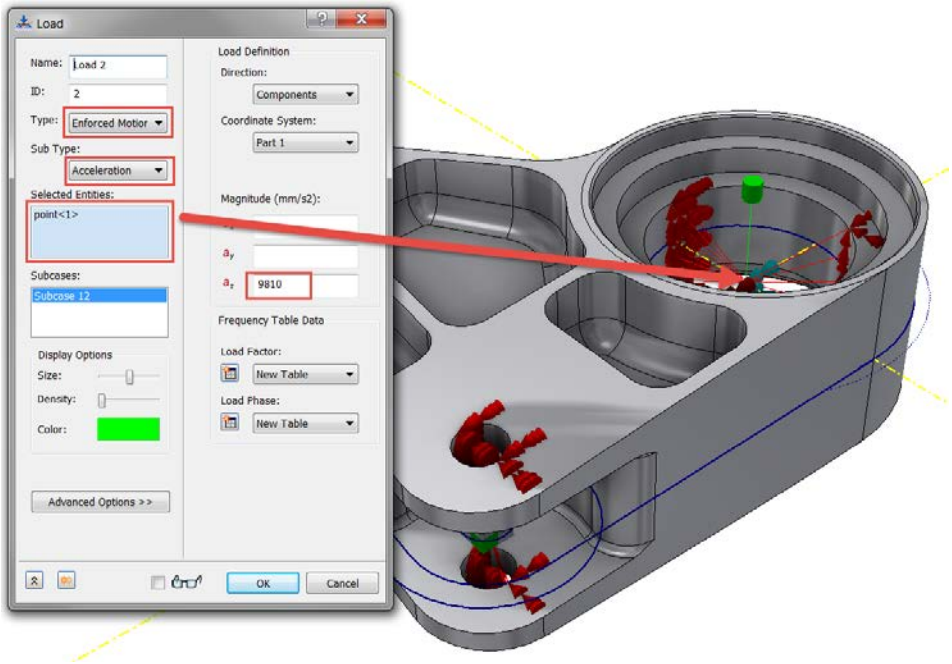
2. Copy the Constraints, Connectors, and Concentrated Mass to the “Frequency Sweep” study.



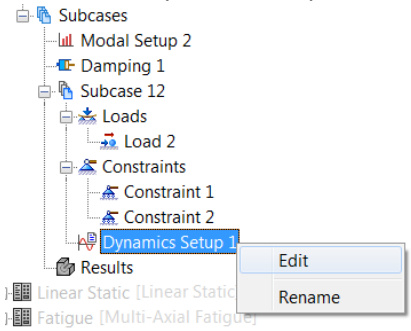

3. Turn on 2D sketches if they are not already on.



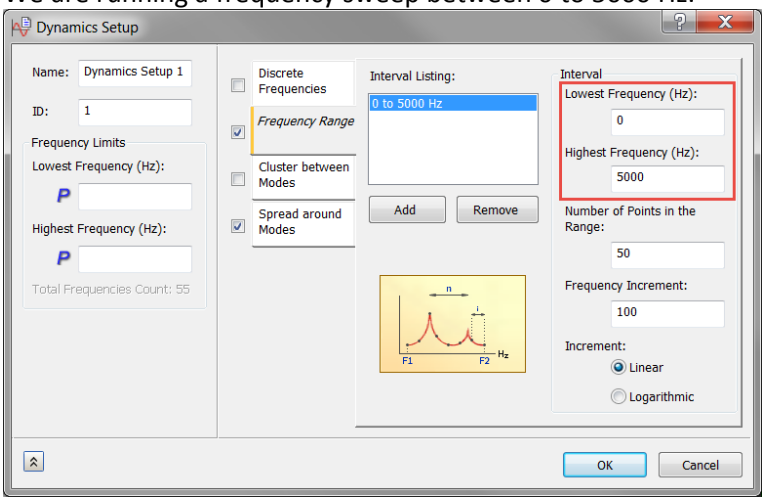
4. Apply a load to the center point of the main bore. This will be an enforced motion that is normal to the road of one G or 9810 mm/s2.



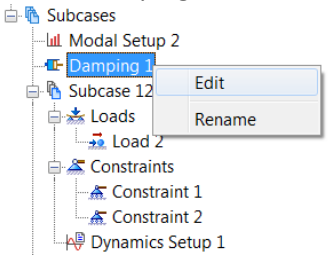

5. Review the Dynamic Setup.

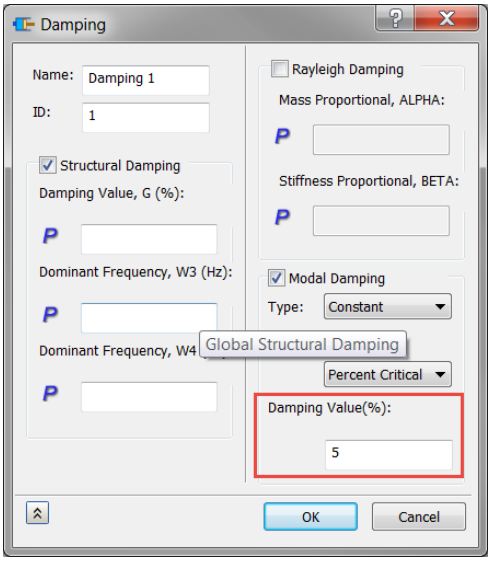


6. We are running a frequency sweep between 0 to 5000 Hz.



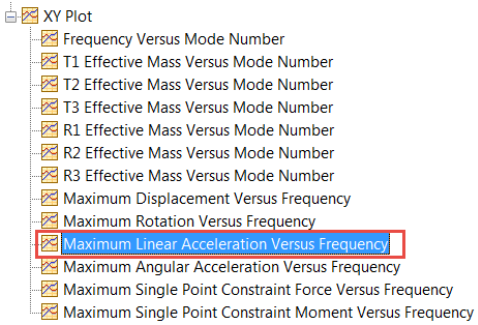

7. Review Damping.



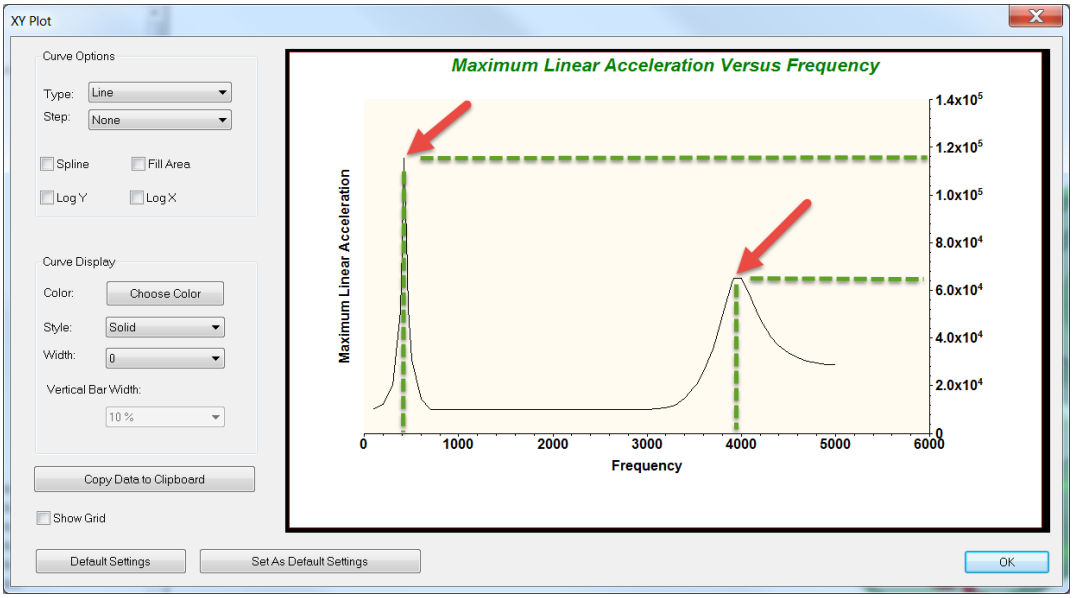



8. Mesh and Run
   *Note: If the study fails. Try deleting the acceleration load and create a new one.*

9. Check the maximum linear acceleration over the 0-5000 range of frequencies.




10. Notice there are two peaks. The first one is at a very small frequency you might see when starting the car. The second one is at 4000 Hz which is probably beyond the range that the BAC Mono operates in.




## Conclusion

I hope you found the information in this class to be helpful to understand the stress and factor of safety results on your components. Once again the handout covered one of several examples used in the course. Check out the class recording and power point for additional information. Feel free to contact me if you have any questions regarding the material.

Jim.byrne@autodesk.com

Twitter – @jimbyrne7777

LinkedIn