# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting

AT12113-L
Andrew Sears
Autodesk, Inc.

## Speaker Biography

*Based in Lake Oswego, Oregon, Andrew Sears is a quality assurance engineer working on Autodesk Nastran In-CAD software. He was designing heavy equipment for the mining and aggregate industries for 6 years before coming to Autodesk, Inc., in 2011. He has spent most of his time designing assemblies with over 10,000 parts, so he brings some practical knowledge of designing large assemblies and creating simulations. Andrew is well connected to the engineering community in the Portland Metro area and has been president of the local Inventor User Group for over 6 years.*

## Learning Objectives

- Learn how to set up and run a Shape Generator analysis
- Learn what the results mean
- Learn how to incorporate the results into your production model
- Validating results with AIP Stress Analysis

## Description

Learn how to use Inventor software to create lightweight versions of your part design. We will have a hands-on lab and you will have time at the end to provide feedback to the Development Team.

## Your AU Experts

***Travis Evans*** *joined Autodesk in 2010 as a Manufacturing Technical Specialist. He is part of a team comprising of AEC/ENI/MFG/M&E experience that is known as the Autodesk Frontline team. In this role he supports the Channel in presales engagements by educating their customers with his knowledge of the Manufacturing portfolio and Autodesk's Digital Prototyping solution.*

*Prior to joining Autodesk, Travis studied Mechanical Engineering earning his degree from the University of Utah. It is there were he first started using 3D design tools as he took part in the Society of Automotive Engineers Mini Baja Series; designing, fabricating, and testing a one person off-road vehicle.*

# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting

## Learn About Shape Generator Model Setup and Analysis

**Swing Arm**

Let's take look at a possible starting point of a cast swing arm with the minimum machining performed. The goal is to find places in the model where material can be removed while meeting the design requirements. This workflow will help you become familiar with the Shape Generator environment.

***User Interface***

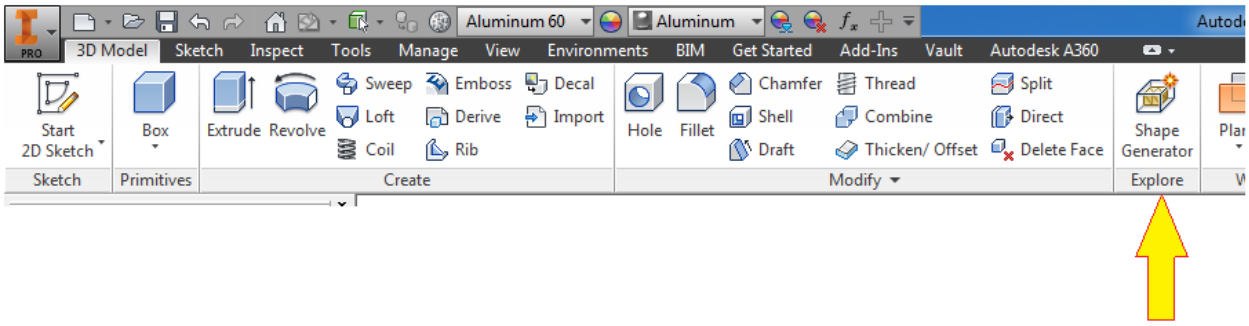
You will take a few minutes to review the Shape Generator user interface.

Step:

Open [**swing arm.ipt**].

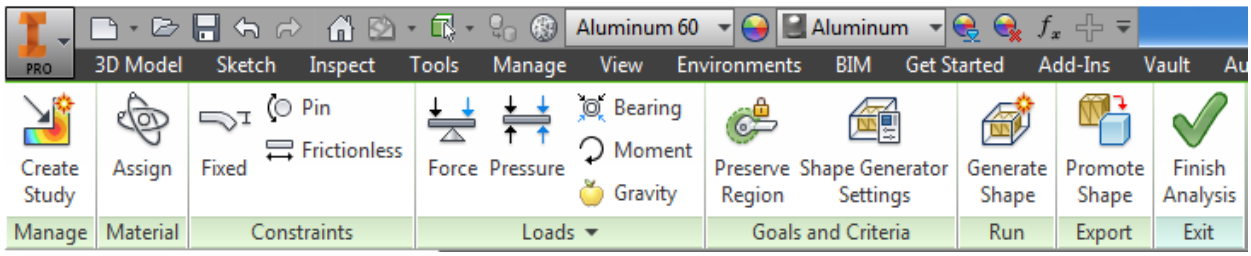
Step:

You will notice a new icon in the 3D Model tab called Shape Generator. Click this to enter the Shape Generator Environment.




The Shape Generator environment will look familiar to AIP Stress users because this is where Shape Generator was added for AIP users. This is also available to AIS users. For AIS users the only thing that is missing is the Create Study Ribbon icon. All other Shape Generator functionality is the same for both versions of Inventor.

Now let's discuss the Shape Generator Ribbon in context of the Panels.




- Mange will allow you to create new Stress analyses that will compliment your Shape Generator data.

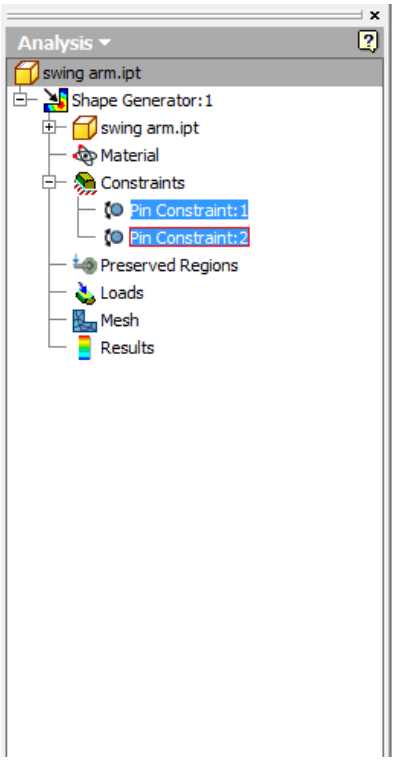
- Assign will allow you to change the material definition as needed. If an invalid linear analysis material is defined, we will default to Steel.
- Constraints will allow you to define how the component will be help during the analysis. We do not allow Free-Free conditions so a fully constrained model is required.
- Loads will allow you to define the forces on the model.
- Goals and Criteria will allow you to define areas of the model that you do not want material to be removed and Settings will allow you to define a target amount of material to remove along with basic mesh controls.
- Generate Shape will start the analysis and give you a solution to review.
- Promote Shape will allow you to insert a .STL representation of the approved solution to the document as a reference or save an external copy of the .STL representation.

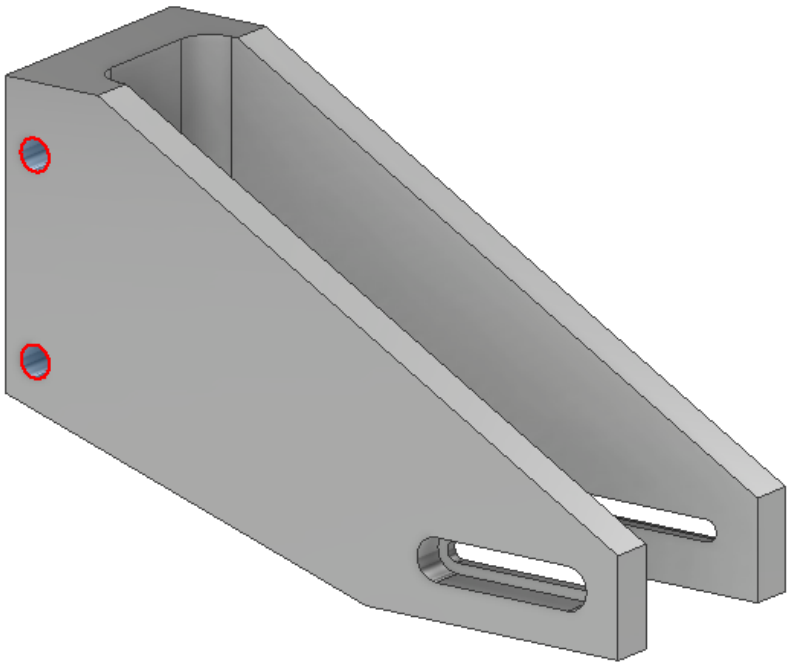
With [**swing arm.ipt**] open, let's create a shape!

***Adding Constraints***

Step:

Add pinned constraints to the holes at the end. To do this, select the Pinned Constraint, choose one hole and select apply then select the other hole then select OK. You should see the two Pin Constraints show up in the Browser.



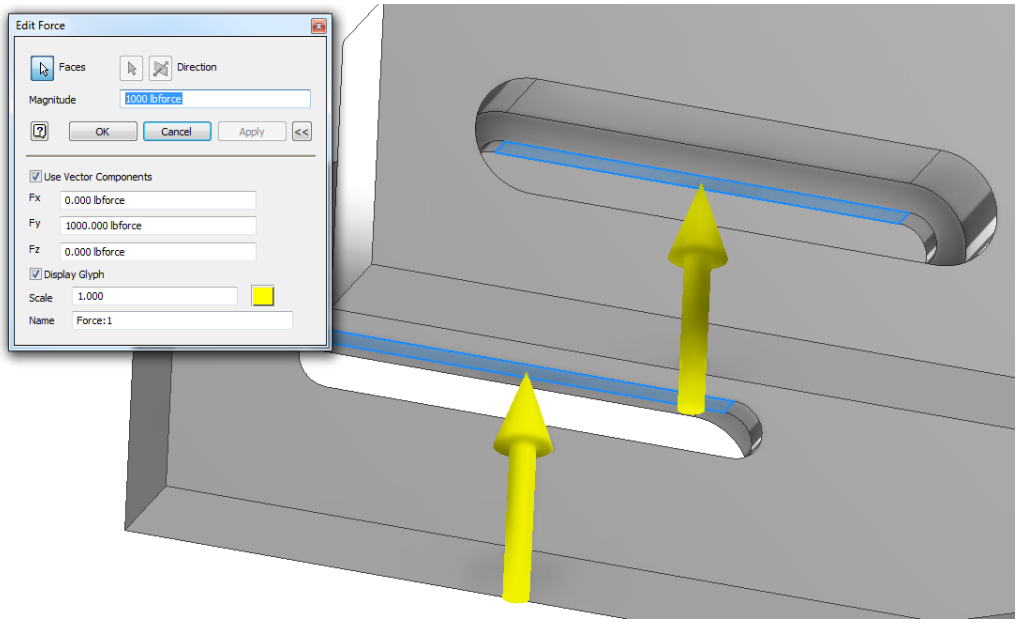



# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting

***Adding Forces***

Step:

Apply a force to the small surfaces in the slotted section. The magnitude will be 1000 lbforce and normal to the surface. This will simulate an upward load on the swing arm. Hit OK to apply the load to both faces.

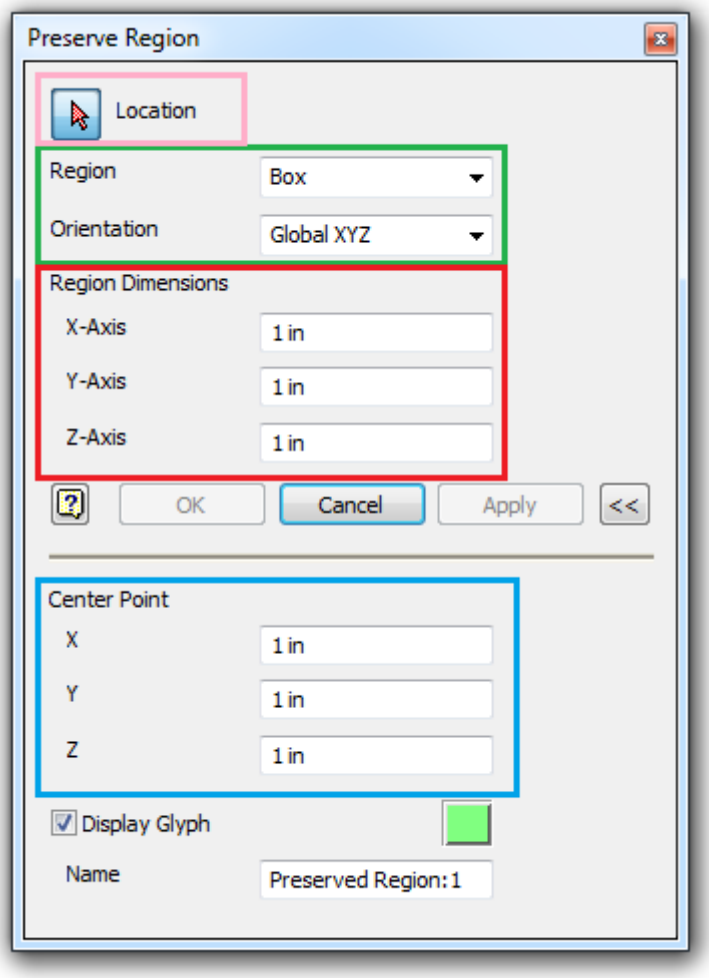



NOTE: By splitting a load on both faces, this means the load is distributed over the areas. For example, let's say you want to define only one load with a magnitude of 1lbf. If two independent surfaces are being considered for loading and one surface represents 75% of the total area then the second surface will represent the remaining 25%. The single applied load will be split at 75% giving you 0.75 lbf and the second surface will have 0.25 lbf. If you wish to have the load equal and independent of the surface area applied, two separate loads will be required with the same magnitude.

***Preserving Regions***

Let's define some regions to preserve and exclude from material removal. Select the Preserve Region icon in the Ribbon and let's review what is available before defining areas on the model to preserve. The Preserve Region dialog can be thought of as four distinct sections shown below. We will discuss this from top to bottom.

# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting




- **Location** will select a start point and also define the Region shape.
- **Region and Orientation** will define the shape used to define the model geometry you wish to keep. You have two choices that are Box and Cylinder. Switching between the two Region types will adjust the Region Dimension inputs. If you choose a flat face, Region will be a box. If you choose a cylindrical face, Region will adjust and change to Cylinder. If this assumption is not what you want, you can change
- **Region Dimensions** control the size of the preserving shape. The Box Region option will use X, Y and Z and the Cylinder Region option will use Radius and Length.
- **Center Point** will control the Region position in the Orientation Global XYZ coordinate systems or Aligned with the geometry you have selected.

Understanding this section will be the hardest part of learning how to use Shape Generator. If you have any questions, please ask one of us. This will be better understood after working through the model.

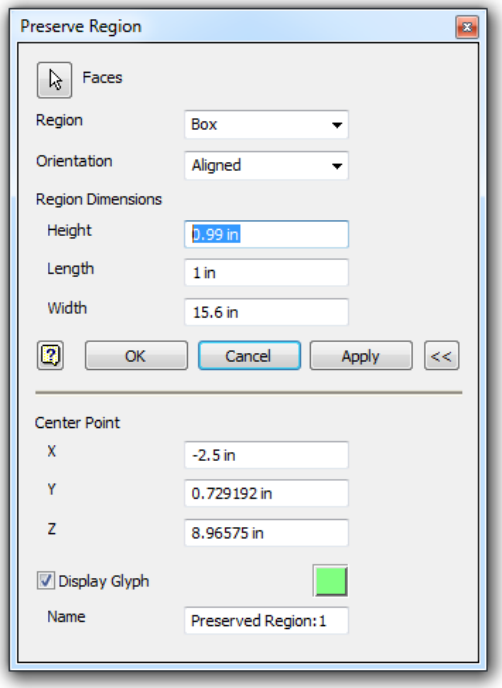
NOTE: Currently, Preserve Region only supports single face selection. Selecting Apply will define the new Preserved Region and keep the dialog open for more definitions.

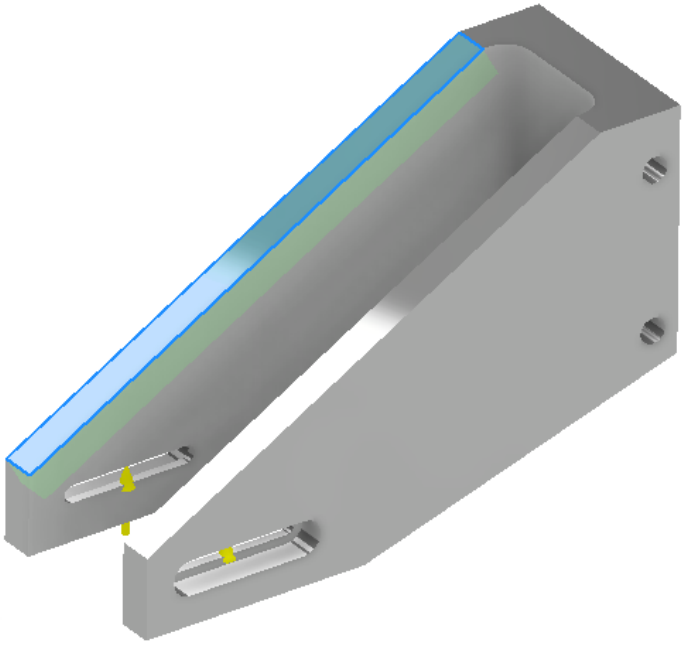
NOTE: All boundary conditions and loads will have a one element thick layer preserved region.

# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting

Step:

To start with, let's define the regions to preserve for a couple faces. The steps will walk you through defining regions to preserve at the top, sloped faces then moving to the underside, sloped faces. Start with Box Region. Pick the top face shown below.



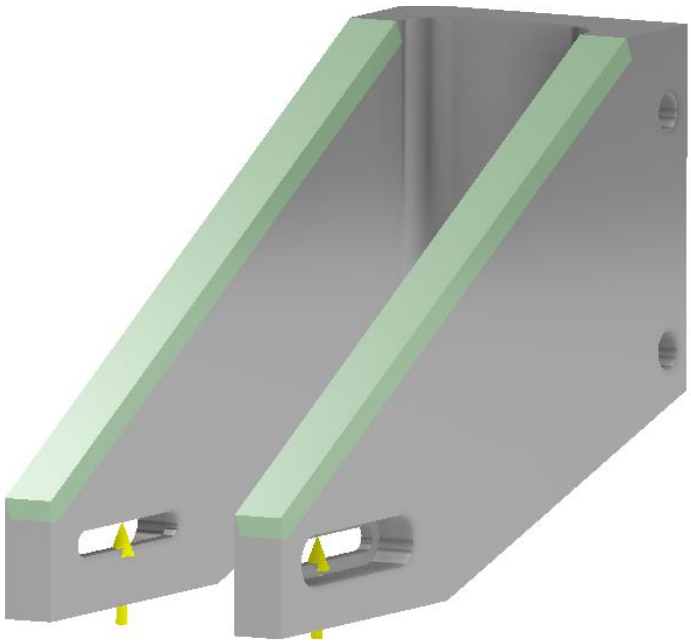



The Height Dimension is normal to the surface. Change this to 0.5 in and select Apply.

Step:

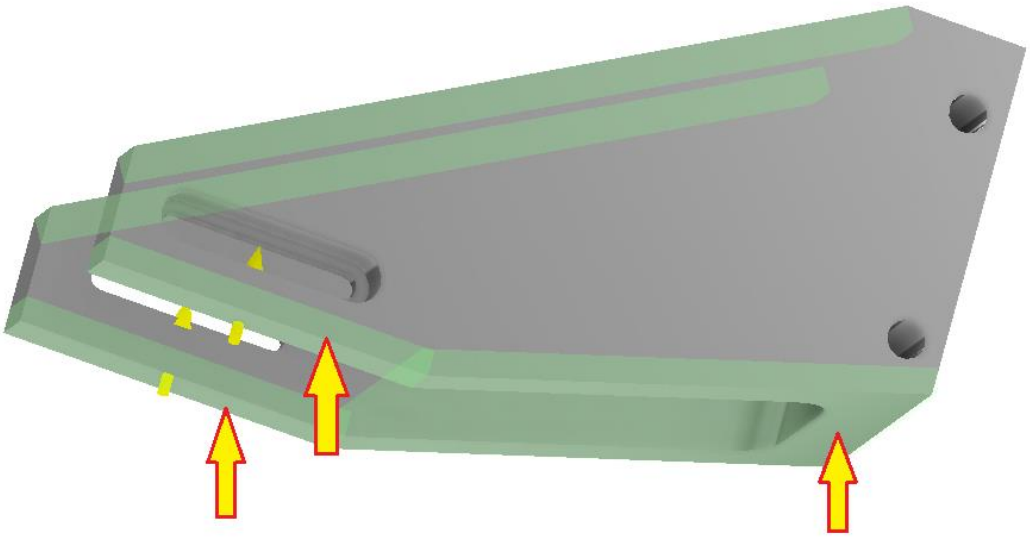
Select the opposite face and change the Height to 0.5 in and select Apply. Your model should look like the one below now.

# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting



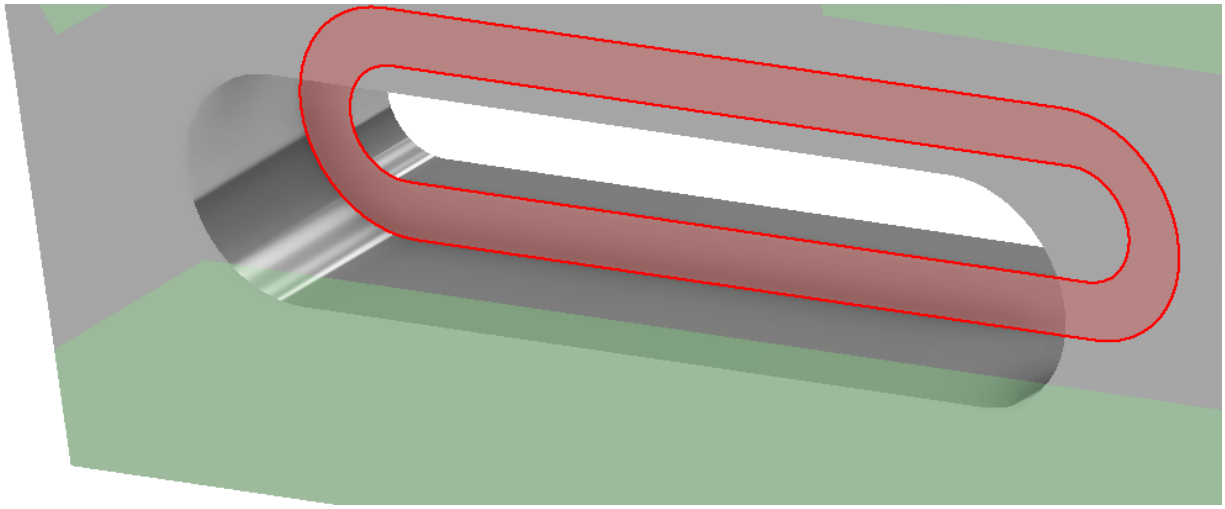
Step:

Add Preserved Regions as shown in the image below. Set the Height as 0.5 in for all three faces.



Step:

Now let's preserve a region that what will be based on one face but will cover multiple faces. At the slotted section, choose the inside face as shown below. For clarity, I turned off the Force glyphs.



NOTE: This will use a Box Region since the face selected is flat and the Orientation is set to use Aligned.

Step:

Modify the Height by inserting 6 in. This will extend the Preserved Region preview through the part.

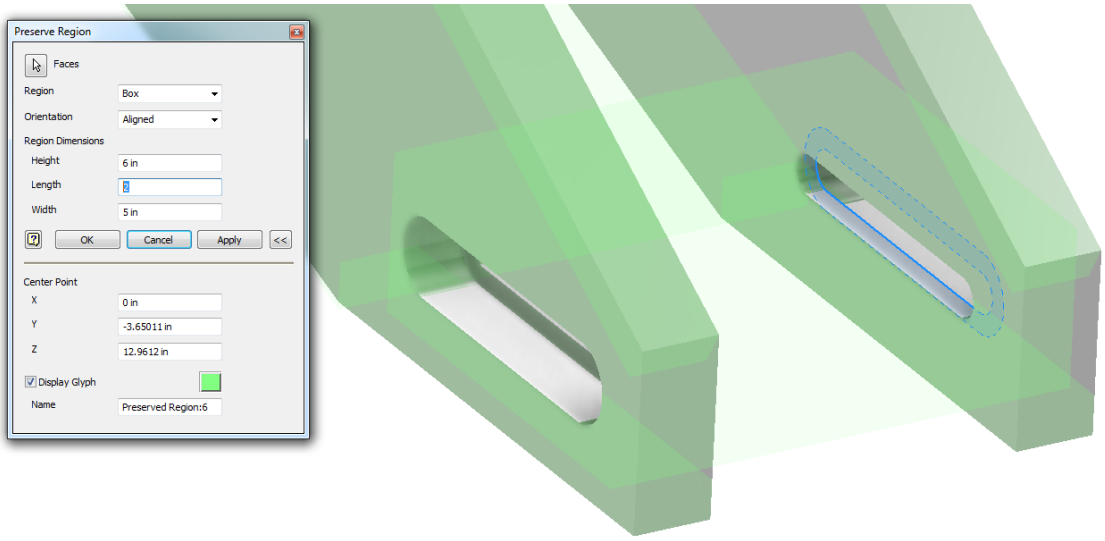
Step:

Change the Width to be 5 in and the Length to be 2 in.

NOTE: Width and Length are adjusted from the center outward as height is measured normal from the selected face.

Step:

Adjust the X value in Center Point section to be 0 in. This will slide the Box to cover all model geometry around the slots. Your Box should look like this below.

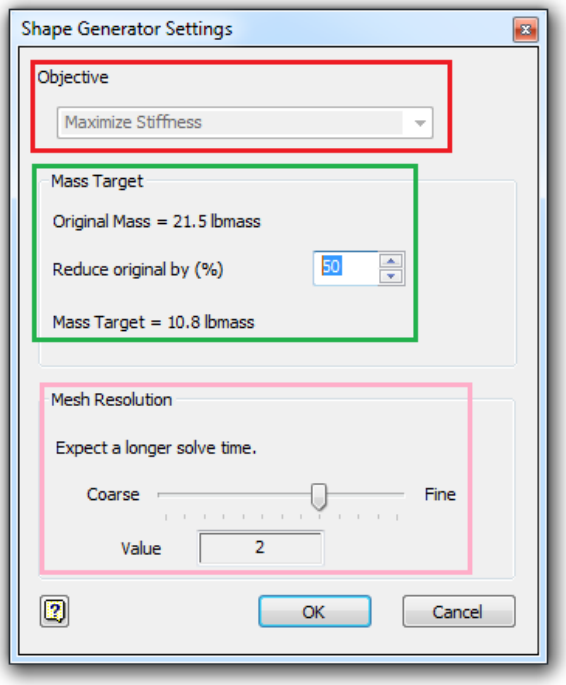



Step:

Select OK to apply the changes and close the dialog. Save your model.

***Shape Generator Settings***

The Shape Generator Settings dialog has three sections. Objective, Mass Target and Mesh Resolution.



- **Objective** will define the criteria for the solver. The solution is currently limited to maximizing stiffness of the structure but we are researching other options.
- **Mass Target** is used to define the amount of material you wish to reduce the model by.
- **Mesh Resolution** will give a simplified way of deciding the density of the mesh. This is controlled by a scale of 7 (coarse mesh) to 0.5 (fine mesh). This is not available for viewing until a solution is found first but we are researching other options.

Step:

Set the Mass Target to 50% and the Mesh Resolution to 2 then select OK.

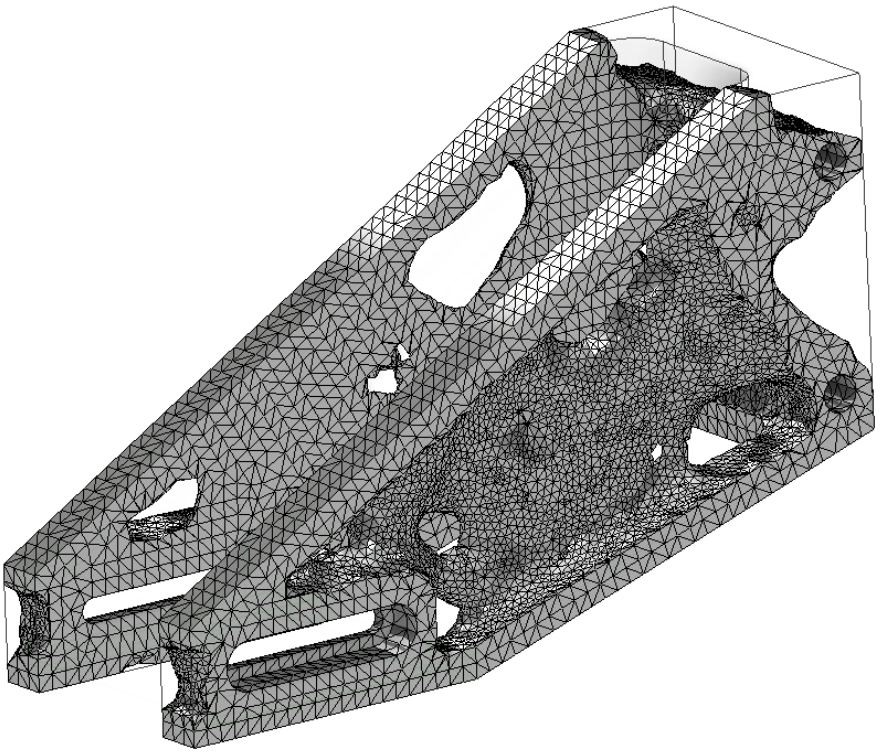
***Generating a Shape***

Step:

Select Generate Shape. This will take about 3-5 minutes to solve depending on the computer.

NOTE: You might see WARNING T2299: REMOVING ISOLATED ELEMENT ### FROM OPTIMIZATION DESIGN SPACE. This is to let you know that at some point during the iterations, an element was separated from the rest of the mesh. Since this does not help in your solution, we are automatically removing these floating elements from you result.

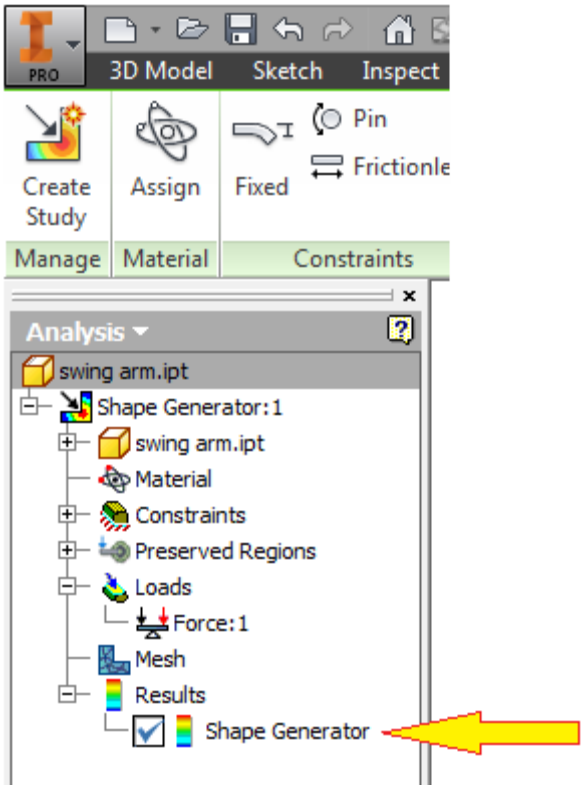
Here is the shape I got. Share your results with us if you have something drastically different.



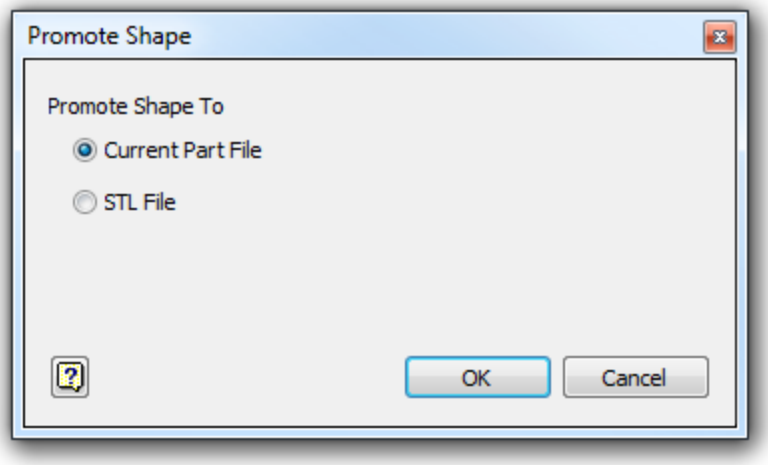
# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting

***Promoting the Shape***

Now that you have a shape let's talk about what you can do with this. At this point the shape only exists as a Result in the Shape Generator environment. To save this shape as .STL data in the model environment you can use Promote Shape.

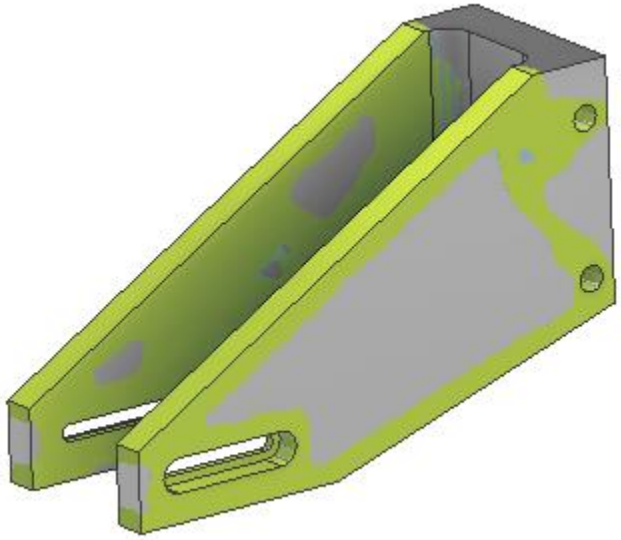



Promote Shape has two options. You can insert the .STL data into the model or as an external file. For this example you will import the data into the existing file.




Step:

Select Promote Shape form the Ribbon and choose OK. This will save the mesh data to the model and take you out of the Shape Generator environment.



NOTE: Mesh data is stored in the Browser at the end of the feature tree and is Yellow for easier viewing.

***Modifying the Geometry***

Now that the Mesh data is available in the part file you can use this as a reference to edit the solid geometry. It will be important to see the imported mesh data without the original CAD data covering it up so you can start removing material.

Step:

Change the modeling view to Wireframe. This will make the CAD model transparent with the Mesh data opaque.

Step:

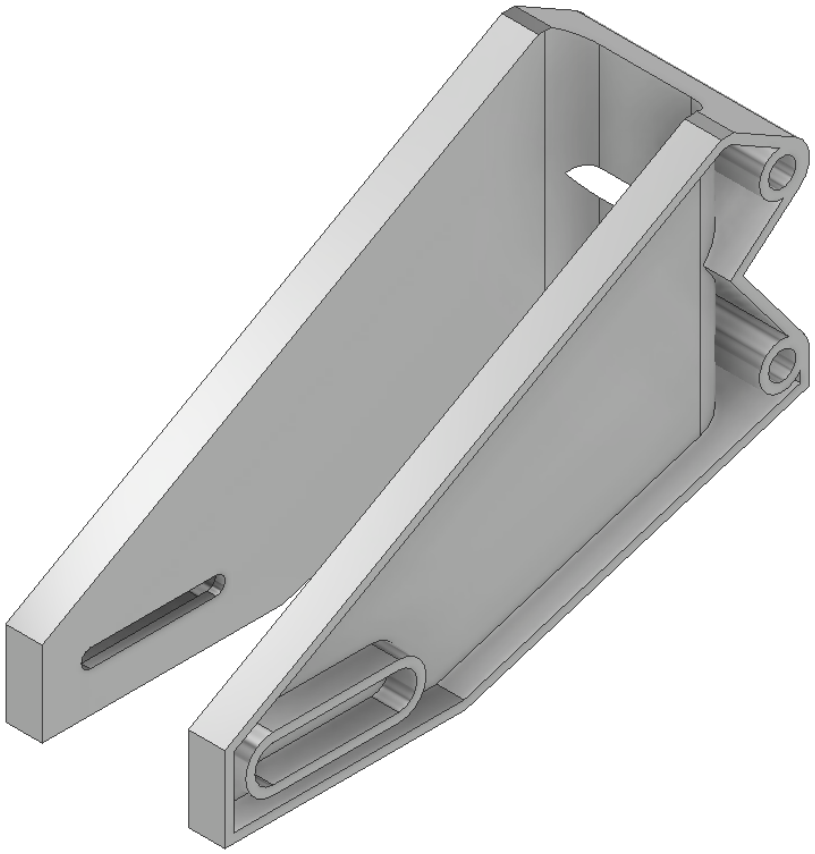
Start a sketch on the side face of the part. This will be the starting point for removing material. Below is an example of what did.

Step:

Start creating the extrusions to remove material for areas that go all the way through the part. You can use lines, circles, rectangles, etc. to get to the end goal. Play around with different options and see what you like best.

Step:

Use the Shell Modeling command and select both outside faces then put in 0.25 for the thickness. The Shape Generator solution shows that you can remove material from the outside but can leave the inside wall.



Now that you have a slimmed down model, you should now verify this shape is satisfactory with a stress analysis.

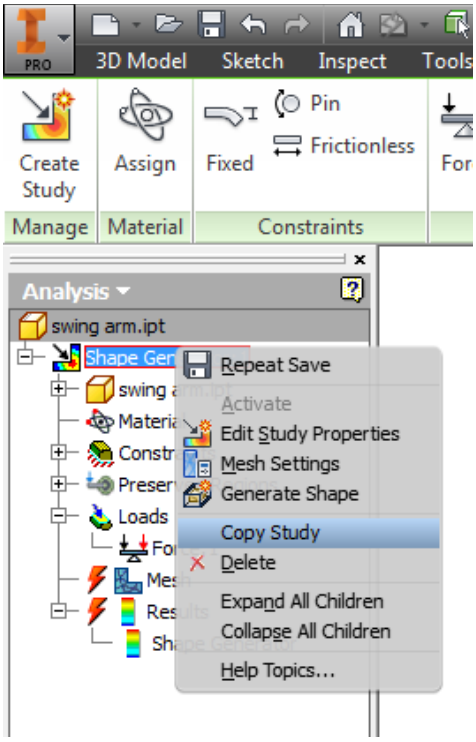
***Stress Analysis***

Step:

Now that you have made edits to the model, go back into the Shape Generator environment.

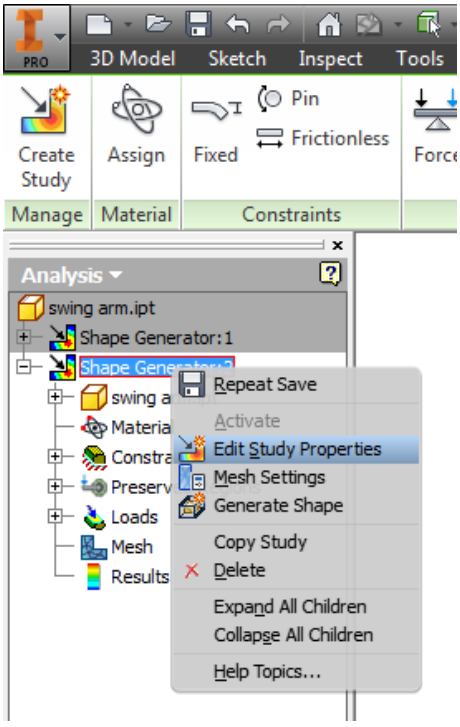
Step:

RMB on the Study and choose Copy Study. Refer to the image below.




Step:

RMB on the copied Study and select Edit Study Properties.



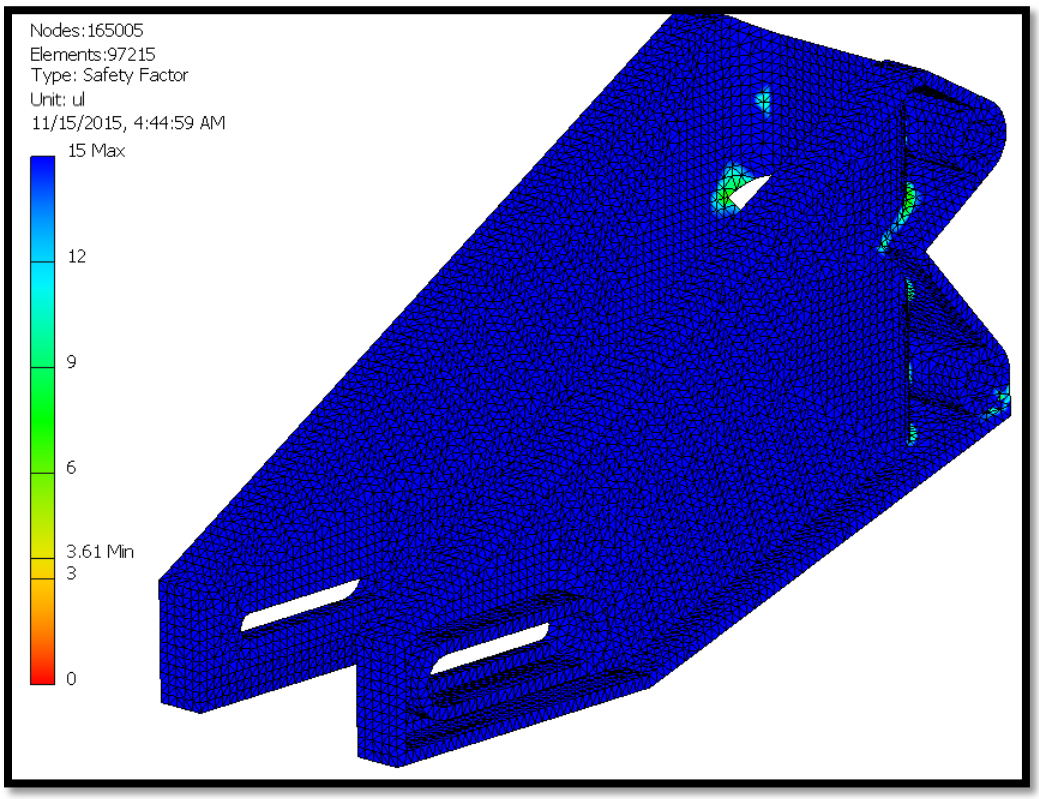
# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting

Step:

Change the Study Type to Static Analysis and select OK.

Step:

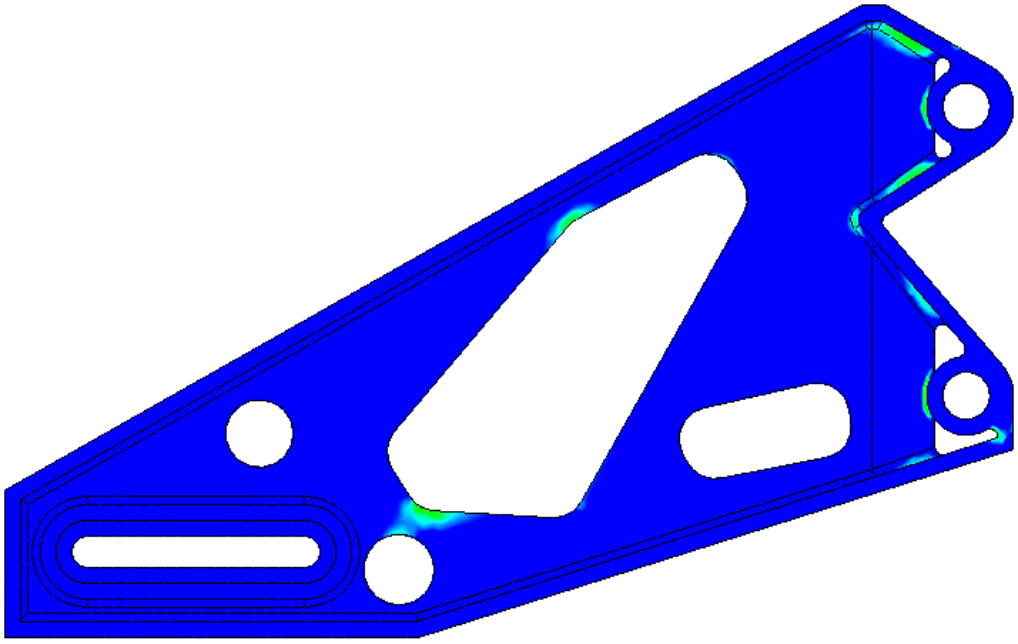
Run the Static Stress analysis.




Conclusion:

The original part started out as 21.5 lbf and the Shape Generator solution gave suggestions on where to remove material and achieve the 50% weight reduction. My first iteration of the Shape Generator solution resulted in 8 lbf and a Safety Factor or 3.6. The Shape Generator solution showed that even more material could be removed from the inner faces and the Stress Analysis results support this. With more refining, here is what I was able to create and reduce the weight to 7 lbf.

# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting



# Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting

This will conclude the Lab. Thank you so much for taking this lab and providing any feedback to us in the room. If you have any questions after AU about what I created for you, please email me at andrew.sears@autodesk.com.

Thanks again!

Andy Sears