# Evolved Structures Guide for GSFC Applications

Ryan McClelland, NASA Goddard Space Flight Center, Code 550

## 1 Introduction

### 1.1 Overview and Motivation

For an introduction to the motivation and technology behind Generative design and Digital Manufacturing, please watch: Ryan McClelland – NASA - Generative Design & Digital Manufacturing at NASA Goddard - CDFAM (youtube.com)

This guide enables optimized structural parts to be designed, validated, and prepared for manufacturing quickly and efficiently. For simple parts with known requirements, designs can be completed and validated by Finite Element Analysis (FEA) in as little as 1 day by an experienced user. This process is tailored to GSFC applications and standards.

The recommendations in the guide should be considered *smart defaults* and are superseded by project-specific requirements.

### 1.2 Software

Creation of **Preserve** and **Obstacle** geometry can be completed in the user's CAD tool of choice (e.g., Creo, SolidWorks) then exported/imported as a STEP model into a generative solver of their choice; this guide focuses on Fusion 360. **Preserve** and **Obstacle** geometry can be separate bodies parametrically tied to the native CAD. Only the geometry recommended in this guide should be imported into Fusion 360 to keep the Generative Design models as small and simple as possible. Assemblies such as detectors and optical systems should be represented as volumes and interfaces rather than wholly importing them. Care should be taken to align the reference coordinate system used for export with any desired machining axes, since Fusion 360 doesn't currently support multiple coordinate systems. Once in Fusion 360, imported geometry can be moved, rotated, scaled, cut, re-sized, mirrored, etc. within Fusion 360, which includes a full-featured CAD package. **Preserves** and **Obstacle**s can also be created natively within Fusion 360.

Generatively Designed part can be most easily modified within Fusion 360 due to the extensive T-spline editing tools for organic shapes, then output to the user's preferred CAD via STEP for additional modification if needed.

#### 1.2.1 Installing Fusion360

This guide is specific to the Autodesk Fusion 360 Generative Design software but may be updated as other tools come into use.

Download Fusion 360 for Free | Free Trial | Autodesk

This guide assumes you have familiarity with Fusion 360 Generative Design. If not, please complete this tutorial first. This guide does not include click-by-click instructions. Fusion 360 selections, commands, and options are denoted with **bold text**. For general Fusion 360 training (e.g. not Generative specific) check out Fusion 360 fundamentals. Lars Cristensen's Youtube channel is also a popular user resource.

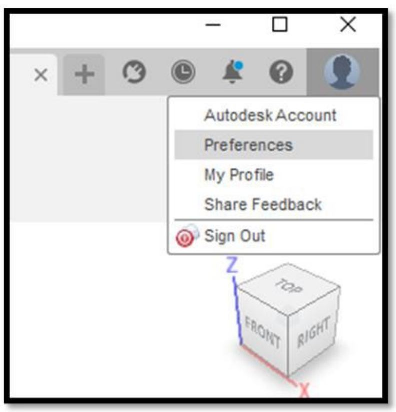
When opening/importing external CAD geometry in Fusion 360, immediately turn on **Design History** so changes are captured in the hierarchy.

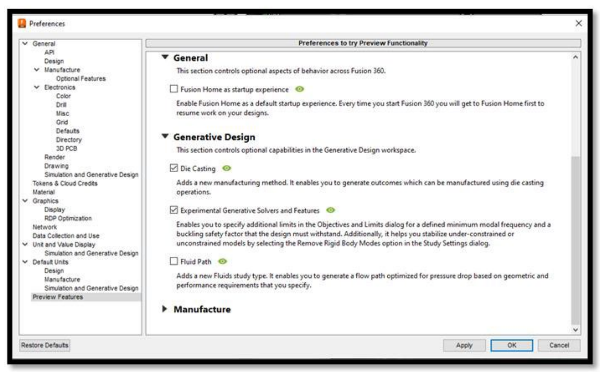
### 1.2.2 Enabling Advanced Features in Fusion 360 (Optional)

Certain features used to require Advanced Features to be enabled in Fusion 360; any of the features referenced in this guide are now incorporated into the main software branch. The following information is included for historical purposes:

There are advanced features in Fusion 360 Generative Design that are considered experimental and require being a part of the "Insiders Program." If you want to use these features:

1. Install Fusion360.
2. Join the Autodesk Insiders Program using the same email you're using Fusion360 with: [https://www.autodesk.com/campaigns/fusion-360/insider-program](https://www.autodesk.com/campaigns/fusion-360/insider-program)
3. Open your account preferences and select Preview Features in the left pane and select" Experimental Generative Solvers and Features" (It may take up to 24-hours for the options to show)




## 1.3 Help and Support

Help is available within Fusion 360 via the help icon in the upper right and the support icon in the lower right. The help icon takes you documentation and the learning panel with context sensitive help. The support icon will allow you to search documentation and learning resources or directly chat with a support agent.

More information on Generative Design, including discussion/help within the GSFC community can be found on the Generative Design MS Team. Please request permissions at the link.

## 1.4 Evolved Structures through the Mission Lifecycle

Structure requirements evolve throughout the project lifecycle, especially for larger missions. The Evolved Structures process is well-suited to this iteration; once the requirements are defined as described in the sections below, generating new designs based on new or updated requirements is much faster than the traditional approach.

Figure 1-1 shows how Evolved Structures fit into the project life cycle. The design is iterated as requirements change and system-level design and modeling are refined. This is the same as for traditionally designed components, but Evolved Structures can be iterated much faster, foster faster system development.

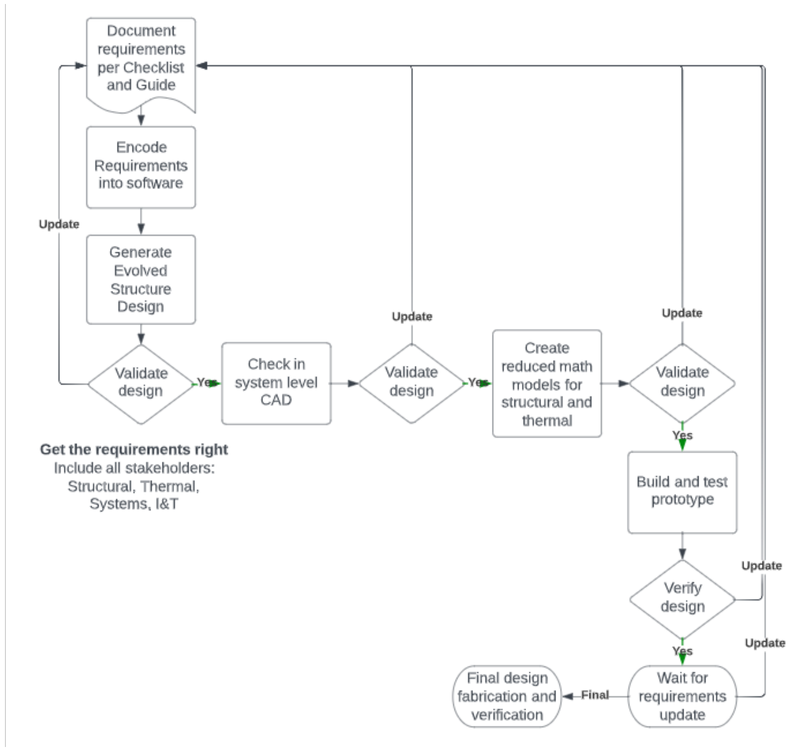



*Figure 1-1. Evolved Structures are iterated through the project life cycle as requirements are changed and refines.*

# 2 Encoding design requirements into Generative Design software

Requirements for the design are encoded into the Generative Design study. First document all the known requirements including interfaces (bolt patterns, bond areas, keep-out zones, clearances), structural loads (force, moment, pressure, g loaded masses), displacement constraints, minimum first mode, thermal requirements, and potential materials. See the Check List on the Generative Design MS Team. Also document the source of the requirement for future reference.

Dealing with unknown requirements such as bolt locations, loads, and modes is addressed in this section. For efficiency in Generative Design, all required **Preserves** and **Obstacles** should be documented and modeled. Otherwise, the model needs to be fixed and re-run.

## 2.1 Bolted Interfaces

### 2.1.1 Finding Optimal locations for bolted interfaces

Early in the design cycles, if bolt locations are not already specified, Generative Design can be used to find optimal locations which will increase stiffness and reduce stress.

Recommended process:

1. Create a large **Preserve** in areas where bolts can potentially be located and apply a fixed constraint
2. Run a Generative Design study with a **Minimize Mass** objective
3. Locations where large structures grow are good bolt locations for a stiff and strong design
4. More than one fastener may be used at each optimal location for redundancy or to reduce the size of the fasteners needed as shown in Figure 2-2.

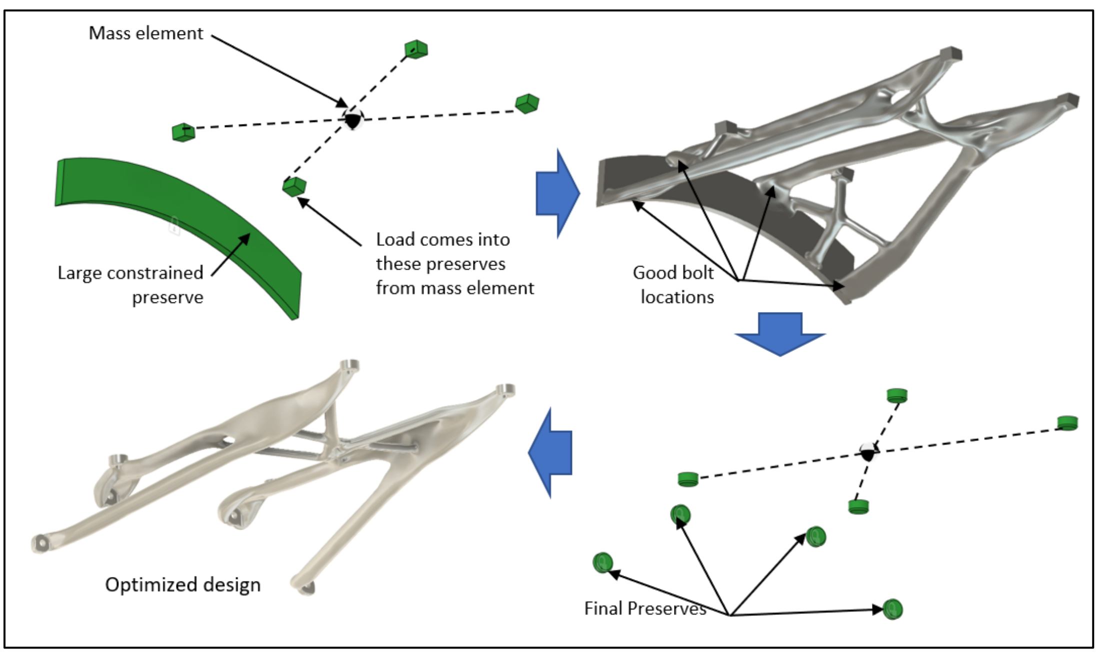



*Figure 2-1. Generative Design Study to locate bolts starts with a large, constrained preserve at the bolted interface (upper left). Areas where large structural members grow are good bolt locations for a stiff and strong design (upper right). Bolt **Preserves**/locations of final design informed by Generative Study (bottom).*

### 2.1.2 Bolted Interfaces at known locations

This guide promotes bolted interfaces for Generative Design with the following characteristics (see Figure 2-2 example):

1. Maintains clearance for bolts and washers considering machining radii and tolerances.
2. Allows material growth around fastener to limit local flexibility, increasing part stiffness and strength.
3. Reduces local stress due to thin sections and sharp corners.
4. Encourages good blending of Generative Design organic geometry into bolted interfaces.
5. Compatible with NASA/GSFC standards for fasteners.

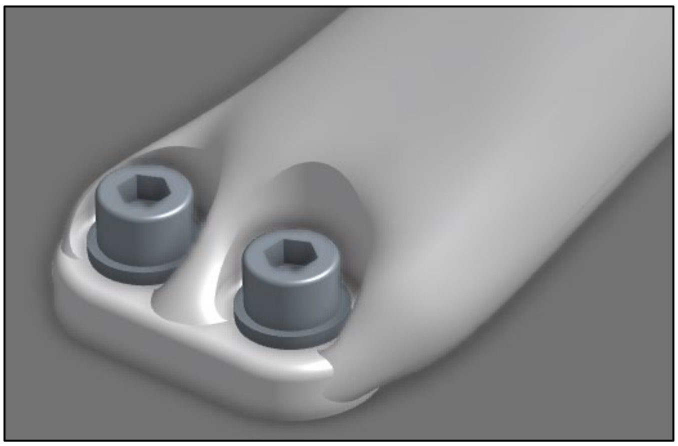


*Figure 2-2. Example of a bolted interface for a Generatively Designed structure.*

Bolted interfaces are represented with both **Preserve** geometry representing the clamped material and **Obstacle** geometry representing clearance for the bolt/washer and installation.

#### *2.1.2.1 Clearance hole preserve geometry*

Rounded rectangular or circular preserves can be used as seen in Figure 2-3 and Figure 2-4. The following **Preserve** geometry is recommended for clearance holes in the *absence of other requirements*.
**D = fastener nominal diameter where in a ¼-20 fastener D = 0.25"**

**Exterior Preserve Dimension = 3 * D (**Satisfies1.5D edge distance req. per NASA-STD-5020 sec. C.2)
**Preserve Thickness = 1 * D**
**All non-flat interfacing edges filet radius = D / 2.5**

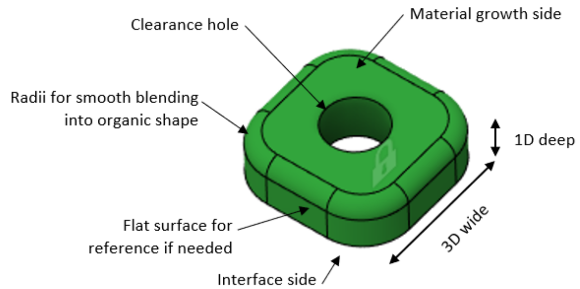



*Figure 2-3 Recommended bolt **Preserve** for clearance holes.*

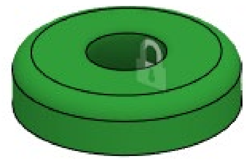


*Figure 2-4. Round Clearance hole bolt **Preserves**.*

### 2.1.2.2 Threaded hole preserve geometry

The following **Preserve** geometry is recommended for threaded holes in the *absence of other requirements*.

- Rounded rectangle or disk 3 * D across like the Clearance hole (Section 2.3.2.1). See Figure 2-5.
- **Preserves** should generally be modeled as blind holes as shown in Figure 4, unless there is an **Obstacle** to ensure the tapped hole will be a thru hole. **If the tapped hole is not modeled as blind, and material grows over an open hole, the design will not reconstruct properly.**
- **Preserve** depth for a threaded blind hole (most cases):
    - 2D depth generally leaves sufficient room for tapping or helical insert installation with 1D thread engagement.
    - Blind hole can be 1.5D deep, but drawing hole call-out will determine exact hole geometry
- **Preserve** depth for a threaded thru hole:
    - If the material is directly threaded, 1.5D deep gives good thread engagement
    - If an insert will be used, the depth should be at least {insert length} + {1 thread pitch}
        - E.g. ¼-20x0.25 Lg Helical Insert = 0.25+1/20 = 0.3” deep
- For directly threaded holes, model the hole as the drill diameter (See Table A8-1)
    - This ensures the hole can be tapped if the part is fabricated to the CAD model
    - E.g. for a #4-40 thread, drill size is 0.089”
- For inserts use the drill diameter for the insert planned e.g.,
    - [https://www.stanleyengineeredfastening.com/-/media/Web/SEF/Resources/Docs/Heli-Coil/HC-2000_rev11_web.pdf](https://www.stanleyengineeredfastening.com/-/media/Web/SEF/Resources/Docs/Heli-Coil/HC-2000_rev11_web.pdf) Table V and VI.

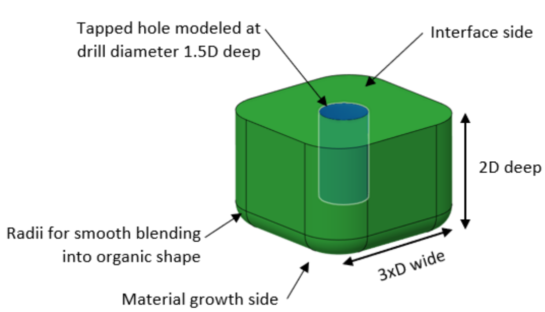



*Figure 2-5. Blind threaded hole bolt **Preserve**.*

#### 2.1.2.3 *Connecting multiple **Preserves** for interfaces in close proximity*

If fasteners are close together (e.g. ~3D-10D apart) connecting the preserves may simplify the design for fabrication and add redundancy to the load path as shown in Figure 2-6. However, this can add mass since the area between the fastener is not optimized.

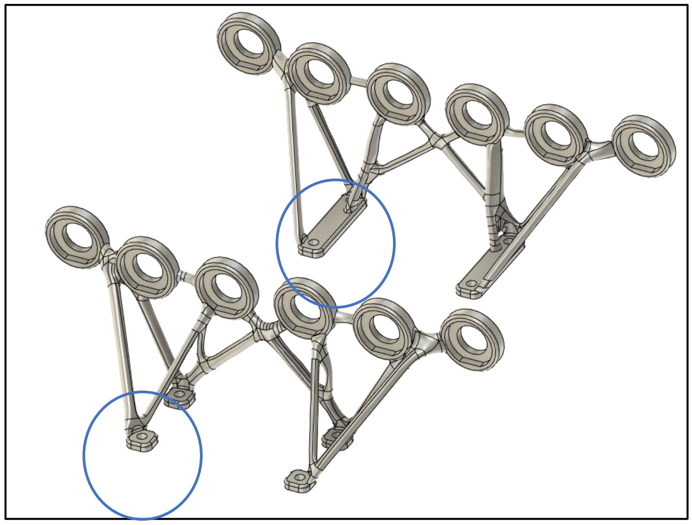


*Figure 2-6. Design with connected preserves (top right) and separate **Preserves** (bottom left). The design with four separate **Preserves** has better stiffness/mass performance but the preserve areas deflected during machining.*

#### 2.1.2.4 ***Obstacles** for bolted interfaces*

Keep-out zones for the bolt and washer must be explicitly defined for each hole. The following **Obstacle** geometry is recommended for bolt holes in the *absence of other requirements*. See Figure 2-7. The **Obstacle** may be created with the built in **Connector Obstacle** tool in Fusion 360, or with extrusions/revolves.

- Clearance or tapped holes should be completely filled
    - Otherwise hole will likely be closed by Generative Design
- **Obstacle** diameter for fastener head should be 0.14” larger than the washer diameter to allow for:
    - 0.125” end mill diameter + 0.014 true position tolerance = 0.14” over washer size
    - E.g. for ¼” fastener with NAS 620 washer, diameter = 0.468+0.14 = 0.608”
    - Washer diameters can be found in hardware specs or vendor data
        - See Table 3 for NAS620 washer diameters
    - For smaller fasteners (e.g. #6 and below) a smaller end mill diameter can be assumed
        - E.g. 0.064” end mill diameter + 0.014 true position tolerance = 0.08” over washer size
- For tapped thru holes, include an **Obstacle** to prevent organic material growing over the open hole, which can cause reconstruction failure.
- Add a radius to the **Obstacle** anywhere material grows to avoid sharp corners in final part
    - For milled parts, this should be the smallest planned end mill diameter
    - For AM parts it is still a good idea to have a 0.064” or 0.032” to avoid stress concentrations
- Depth of the **Obstacle** should be at least 2x(fastener length) to allow for installation
    - If that fastener length is unknown, assume 8D **Obstacle** depth.

        - E.g. 2” depth for a ¼” fastener.
    - Also consider tool access: 15° maximum angle per GSFC Torque spec 540-PG-8072.1.2-A Section 1.3.2
        - If the bolt **Obstacle** gets completely covered by material growth, the installation direction may need to be specifically defined in the obstacle as shown in Figure 2-8.
        - An obstacle may also be needed below the bolted interface to prevent material from growing into the mating part as shown in Figure 2-7.
        - **Obstacle**s are **Bodies** in Fusion 360 that can be patterned and moved/copied.

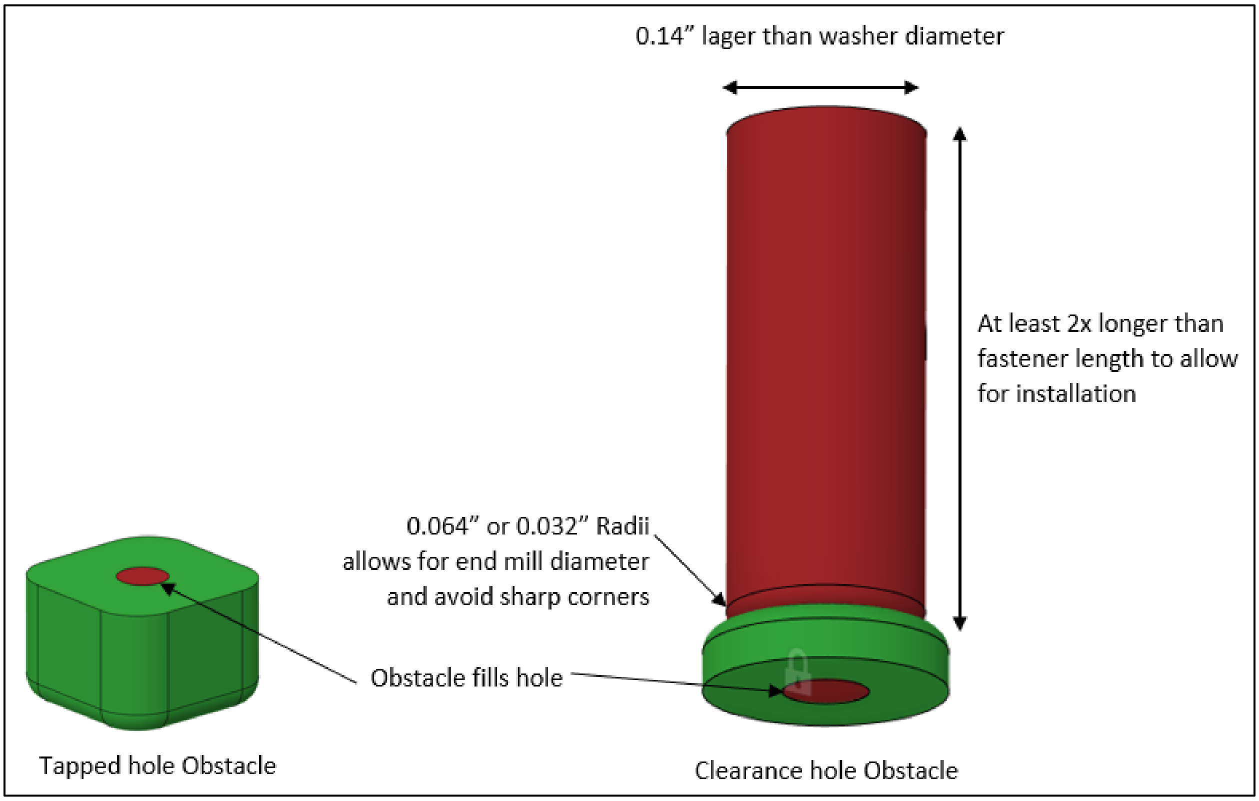



*Figure 2-7. Recommended **Obstacle** for tapped holes (left) and clearance holes (right)..*

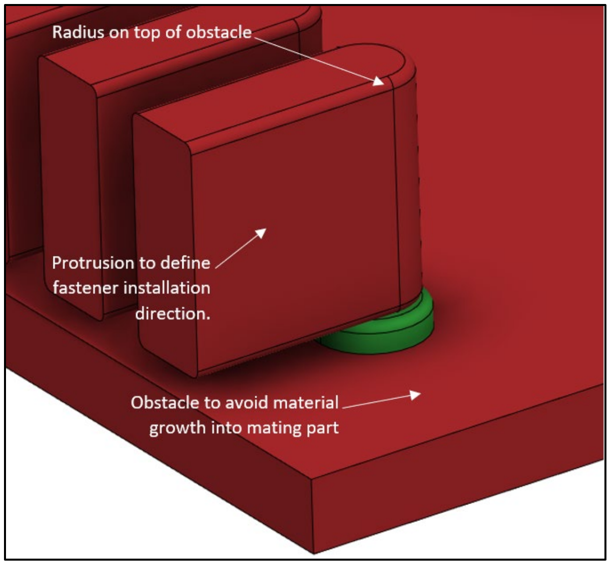



*Figure 2-8. Advanced bolt obstacle when material is likely to grow in undesired locations.*

## 2.2 Pinned Interfaces – Preserves and Obstacles

Pins are often added to bolted interfaces to take shear loads and prevent alignment shift for optomechanical assemblies per NASA Gold Rules (GSFC-STD-1000). **Preserves** and obstacles for pinned interfaces are similar to bolted interfaces, except bolt head and washer clearance is not needed for the **Obstacle**. Also, many designs use a pin and slot combination for easier assembly; slots should be represented in both the preserves and obstacles as shown in Figure 2-9. Pins are usually placed near bolts, so a combined preserve is desired.

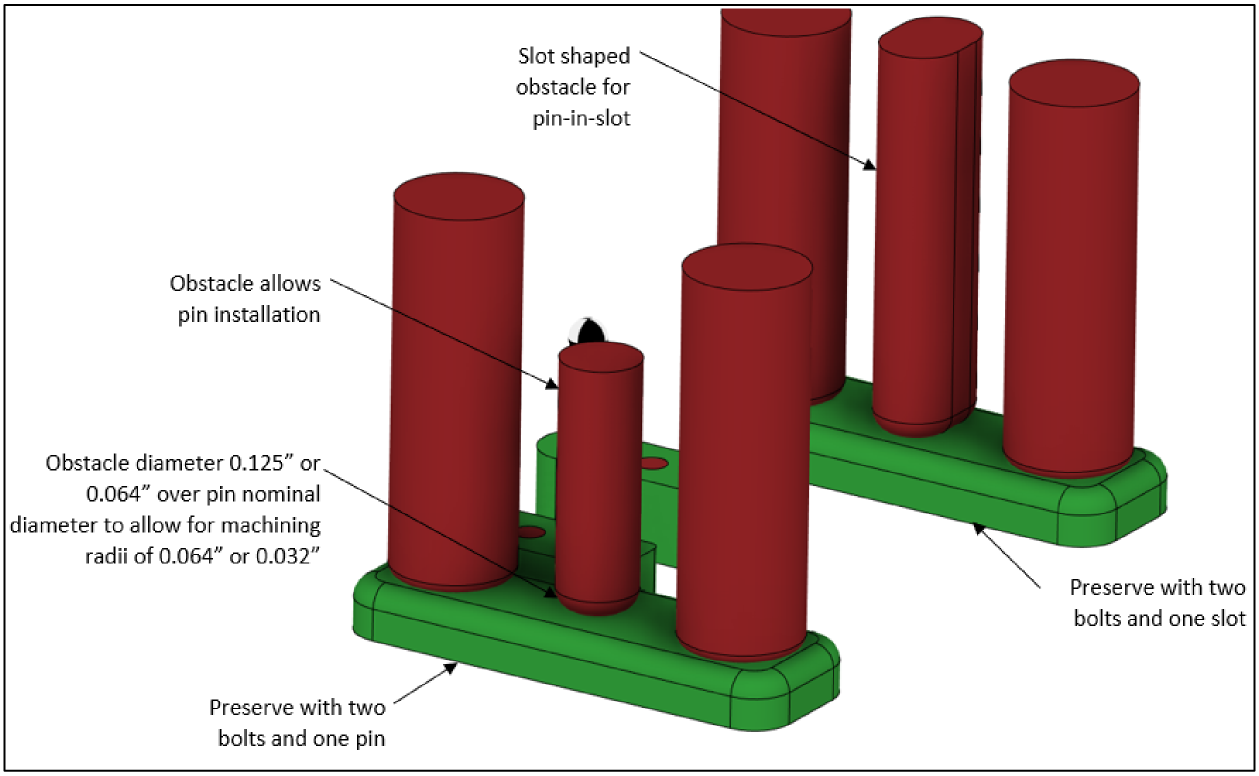



*Figure 2-9. **Preserves** and **Obstacle**s for pins are similar to bolts, except bolt/washer head does not need to be accommodated, and slots are often used.*

## 2.3 Bonded interfaces

Bonded interfaces should be represented by a **Preserve** defining the bond surface and **Obstacle(s)** defining the mating part and any clearance required for injection, clean-up, and inspection.

## 2.4 Thermal considerations

### 2.4.1 Insulating vs. Conducting designs

Structures are usually desired to be thermally insulating or thermally conducting.

Thermal conductance is generally minimized by Generative Design because optimized designs have minimal cross sections. Conductance can be minimized further by using Ti6Al4V and/or including insulating spacers at component interfaces (G10, ceramic, etc.).

When high conductivity is desired, a minimum conductance should be defined which can be used to define a **Preserve** with the desired conductance between two locations (e.g., length and cross-sectional area). This **Preserve** will then carry both heat and structural load. Aluminum 6061 is an excellent conductor and similar to copper.

Also consider separating thermal and structural functions. For instance, Dale Ohm heaters can be used directly on or near components that need to be warmed, or thermal straps can be used to carry away heat rather than relying on conduction through structures.

### 2.4.2 Mounting Heaters, Thermistors, and Blankets

Accommodation of thermal components must be considered along with other design requirements. Thermal component interfaces can be added and updated as the design matures.

The organic shapes resulting from Generative Design can make application of standard Kapton film heaters and thermistors challenging without prior planning. Flat or cylindric surfaces for heater attachment can be added as **Preserves** before Generative Design or added to the part after Reconstruction. The **Cylindrify** and **Flatten** command can also be used to create attachment surfaces on the organic shapes per Section 6.2. As an alternate to Kapton film heaters, compact Power Resistors (i.e. Vishay/Dale-Ohm heaters) can also be added to Aluminum parts; the material's high conductivity allows the heat to spread easily from the concentrated source.

Compared to traditional designs, Generative Designs are often open and sparse. However, thermal blankets can be applied to thermally isolate the structure using GSFC standard processes including G10/Ultem posts or spacers between blankets and structure as well as extra layers at contact points. Properly designed and installed blankets can also be used for stray light mitigation (Section 4), contamination mitigation, and even EMI shielding to some extent.

## 2.5 Other **Obstacle**s

### 2.5.1 Light Path Obstacles

For optical systems, light paths should be modeled as **Obstacle** geometry. Light paths can be simple cylinders and rectangles or complex geometry depending on the application. For a complex light path with multiple obstacles, loft features can be used to capture the light path between optics of different shapes as shown in Figure 2-10. For additional clearance around light paths, **Obstacle Offsets** can be used.

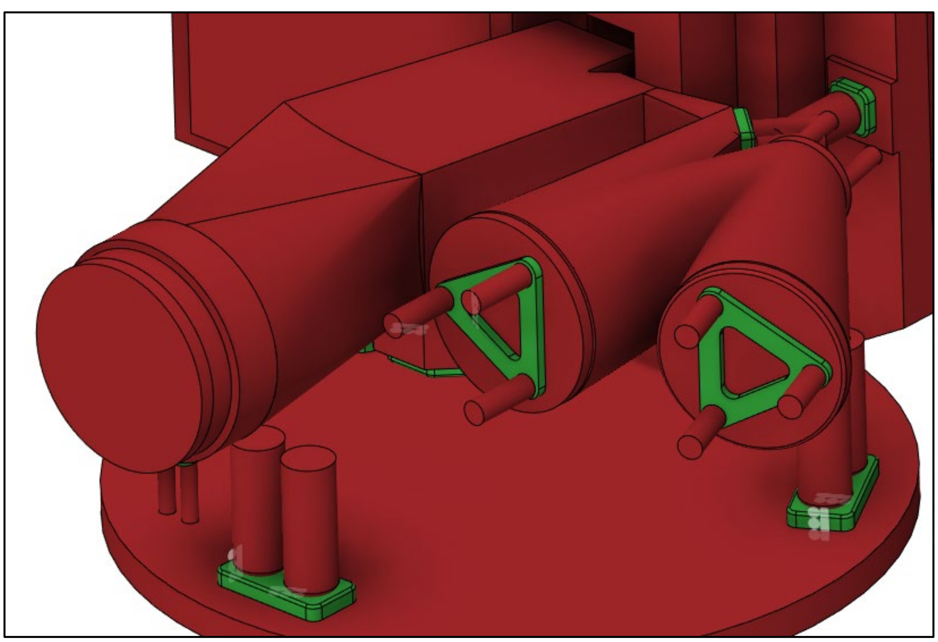


*Figure 2-10. Complex light path **Obstacle**s created with loft features in native CAD.*

### 2.5.2 Non-interfacing component Obstacles

Any component that may fall between preserves should be included as an **Obstacle** with **Obstacle Offsets** as needed for clearance. Radii (0.064” minimum recommended) should be added to the **Obstacle** to prevent sharp corners on the Generatively Designed part.

### 2.5.3 Complex Obstacles Geometry

Fusion 360 can handle very complex **Obstacle** Geometry, however, **Obstacles** should be simplified for faster run time and often better design results (e.g. not overly complex). Within Generative Design > Edit Model, the **Remove Features** tool handles this well. Simplification can also be done in native CAD before importing into Fusion.

All **Obstacle** geometry should be solid or voxels will fill hollow areas leading to longer run times and often bad geometry results. For instance, for an electronics box or detector housing, all internal detail should be removed and any internal voids should be filled a solid extrusion. This can

### 2.5.4 Mounting locations for secondary components

Mounting locations for secondary components such as harnessing, optical alignment aids (e.g., corner cubes), accelerometers, heaters, strain gauges etc., should also be considered. If the component locations are pre-determined they can be added as **Preserves/Obstacles** and loaded with forces/masses or connected to other **Preserves,** so they are not separated/abandoned. Alternately, these interfaces can be added to the Generatively Designed part after reconstruction, particularly if the components are lightly loaded, or their locations depend on the final design.

### 2.5.5 Integration access Obstacles

Generative Design and Digital Manufacturing can be used to consolidate parts for assemblies holding multiple components. However, clearances to integrate parts should be considered and checked. Visualizing the integration via low-cost plastic AM prototypes or Virtual Reality is recommended. If needed, integration **Obstacles** can be created to allow component installation from a particular direction, similar to the bolt installation **Obstacle** shown in Figure 2-8.

## 2.6 Design Conditions

### 2.6.1 Constraints

Constraints represent where the part is mounted at the next higher level of assembly. For bolted interfaces, it is generally conservative to constrain the inside the bolt hole with a **fixed** constraint in all axes. This also allows Generative Design to consider the local flexibility of the bolt **Preserve** and add stiffening material as needed.

Alternately, e.g., if the bolt hole cannot be modeled into the **Preserve** per Section 5.1, the mounting surface at the bolted interface can be constrained.

For bonded interfaces, the bond area should be constrained with a fixed constraint in all axes. For pined interfaces, a pinned constraint should be used.

Flexure interfaces can be modeled with constraints only in the fixed axes or modeled with **Preserves/Obstacles** representing the flexures.

### 2.6.2 Loads

Generative Design will optimize the part across load cases, ensuring requirements are met for each case. No weighting is given to one load case over another.

#### *2.6.2.1 Acceleration loaded masses*

The most common load case for launch is acceleration applied to **Point Masses**. In NASTRAN terms, this is the RBE3 + CONM2 approach typically used to conservatively represent components of unknown stiffness. Each component mounted to the structure should be represented by a **Point Mass** located at the center of gravity (CG) of the component. If the precise CG is not known, an approximate "eye-balled" location can be used for preliminary design. The **Point Mass** is then attached to **Preserve** geometry, usually surfaces where the component is bolted or bonded as shown in Figure 2-11.

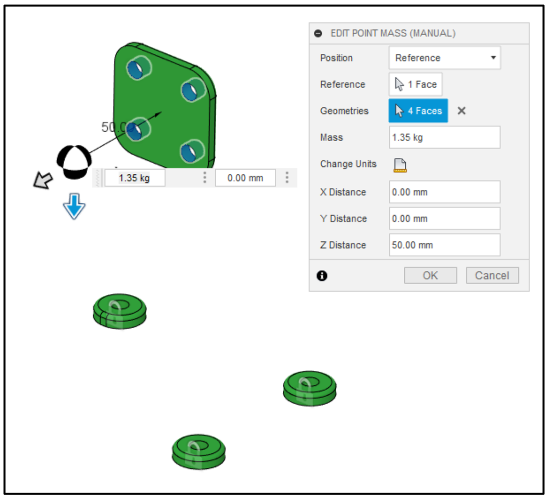



*Figure 2-11. Point Mass representing a Tip/Tilt assembly bolted to a **Preserve**.*

Acceleration loads, also called quasi-static design loads or g-loads, can be supplied by project stress analysts (preferred), or conservatively estimated using a Mass Acceleration Curve (MAC).

For preliminary design, MAC accelerations are applied in the x, y, and z axis independently. MAC accelerations can be developed by summing the Point Masses supported by the part being designed and adding the estimated mass of the new part (20% of the mass supported is usually a conservative estimate) then finding the corresponding acceleration in Appendix Table 8-4. Mass Acceleration Curve (MAC) loads per 542 standards. Linear interpolation can be used between masses.(reference only, based on Figure 2-2/ Table 1 of the Code 542 Guidelines. Alternate link).

([Component 1] 5kg + [Component 2] 10kg)*1.2 = 18 kg ≅ 32g per MAC

A separate load case should be defined for each axis. This can be accomplished by **Cloning** the load cases and changing the acceleration vector. Since the analysis is linear static, sign of the load case does not have an effect.

#### *2.6.2.2 Forces and moments*

Forces and moments may also be applied, for instance from an astronaut kick-load. Generally, these will not apply during launch and therefore should be separate load cases.

Forces may also be used to represent accelerations on masses as described in Section 2.5.1. This may be useful if different components experience different accelerations. In this case, a remote force should be used at the center of gravity with a force of mass * acceleration applied separately in each axis.

## 2.7 Symmetry

If the **Preserves**, **Obstacles**, and loads are symmetric symmetry planes should be defined for the study to yield symmetric parts. Note that the **Preserves** and **Obstacles** geometries must be truly symmetric. Mirror and pattern features can be used to ensure symmetry.

## 2.8 Design Objectives

The preferred **Design Objective** is generally **Maximize Stiffness** because most GSFC structures are stiffness driven and this objective often yields better results. A **Safety Factor** of 2.0-3.0 can be used to start, as Generatively Designed parts usually have low stresses. This also accounts for standard NASA-STD-5001 margin requirements for metallic components and some non-conservatism in the voxel mesh approximation (not using element corner stresses).

The selection of the **Mass Target** heavily influences the outcome and may need to be iterated to achieve a design that "looks good" based on engineering judgement. 20% of the supported component masses can be a good place to start. If modes are high and stress is low, this can be steadily reduced, sometimes as low as 5%. If there is an existing traditional design of the part, 1/2 to 1/3 the mass of the traditional design is usually achievable and makes a good **Mass Target**. Studies with different **Mass Targets** can be run concurrently to speed up the process.

The **Minimize Mass** objective will reduce the mass of the part until the **Safety Factor** or **Frequency/Displacement/Buckling** constraints are violated. This can be useful for identifying **a Target Mass** for a follow-up **Maximize Stiffness** study.

**Frequency/Displacement/Buckling** constraints are not yet well explored for GSFC applications. They can increase run time and don't currently work with all fabrication constraints in Fusion 360. Generally, generative designs easily meet the frequency and buckling requirements when validated with FEA post-reconstruction.

## 2.9 Manufacturing

Multiple manufacturing methods can be selected for the part and will run concurrently.

The anticipated manufacture(s) should be engaged early and asked for **feedback before the design is finalized** in order to reduce delays and cost. Iterating on and verify Design for Manufacturing (DfM) is essential to creating a manufacturable part. Automated feedback can be obtained from manufacturers, but human feedback is needed for AM of 5-axis CNC parts. For both CNC and AM parts, feedback should come from the person who prepares the part for fabrication (e.g. CAM or slicing) and is responsible for the successful build.

### 2.9.1 Unrestricted Manufacturing

The **Unrestricted** manufacturing type should be used on initial runs to see what the optimal load path is absent of manufacturing constraints. This result can also usually be Additively Manufactured.

### 2.9.2 Additive Manufacturing

The **Additive** manufacturing type can be used to design a part built in a certain orientation. Design of Additive Manufacturing (DfAM) is outside the scope of this guide. The defaults for **Overhang Angle** and **Minimum Thickness** are generally good and the **Orientation** should be chosen to minimize the overhangs.

#### *2.9.2.1 Additive Manufacturing Materials*

While material selection is outside the scope of this guide, metal additive manufacturing often defaults to AlSi10Mg; Ti6Al4V is a good choice when low conductivity or lower CTE are needed but costs can vary between the materials. Multiple materials can be selected for each manufacturing type and will run concurrently; custom materials can also be created. Material properties should be checked against MMPDS, but the moduli are generally correct and margins can be written against MMPDS values rather than Fusion 360 material properties.

AM material properties should be taken from the vendor and verified via NASA-STD-6030.

### 2.9.3 Milling

#### *2.9.3.1 Milling Methodology*

Most applications should use the Milling manufacturing type unless AM is needed due to geometric complexity or material availability. A milled part with GSFC's typical materials is the easiest/cheapest to qualify for flight.

3-axis and 5-axis milling can achieve surprising complexity at relatively low costs compared to AM and 2.5-axis milling and 2-axis cutting are available but not well explored for GSFC applications. Milling Design for Manufacturing (DfM) is outside the scope of this guide but it is noted that fewer tool directions generally result in a cheaper but sometimes less optimal part depending on the application. Also, relatively flat applications can sometimes be machined from two axes only, top and bottom. In the absence of other, more specific settings for machines, some recommended settings for CNC Milling are below.

**Tool Direction** specifies which direction the cutting tool will approach from.

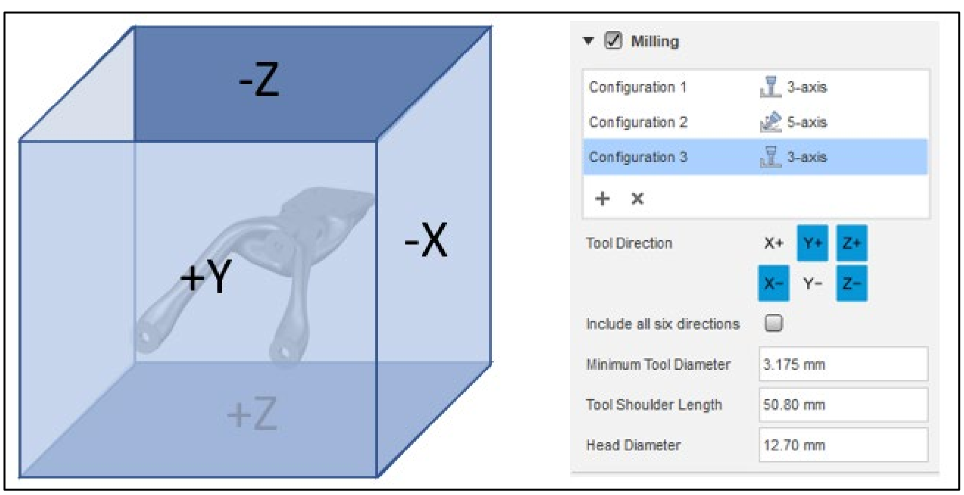



*Figure 2-12. Manufacturers have limitations on machining axes and part sizes. The part shown was machined from 4 axis.*

| Milling Type | Minimum Tool Diameter (in[mm]) | Tool Shoulder Length (in[mm]) | Head Diameter (in[mm]) |
|---|---|---|---|
| 3-axis milling | 0.125 [3.175] | 1.25 [31.75] | 0.5 [12.70] |
| 5-axis milling | 0.125[3.175] | 1.25 [31.75] | 0.5 [12.70] |

*Table 2-1 Recommended Settings absent known specific tooling requirements or capabilities*

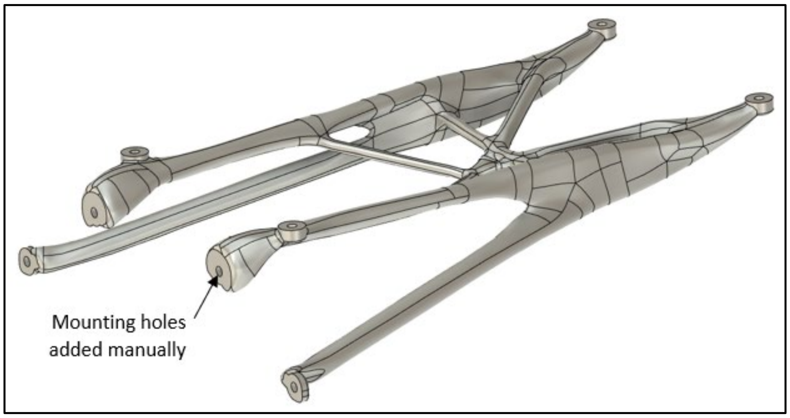



*Figure 2-13. Part milled from top/bottom only. Mounting holes from a third direction were added with manual machining.*

#### 2.9.3.2 Milling Materials

While material selection is outside the scope of this guide. Aluminum 6061 will generally yield the stiffest and most manufacturable design due to its low density, good strength, ease of machining, and lack of warping with when subject to heavy material removal.

Ti6Al4V is a good choice when low conductivity or lower CTE are needed. Machining costs are 3-4x greater than 6061 Aluminum.

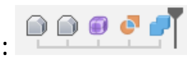
Stainless Steels are generally not recommended for structures due to their high density.

# 3 Fusion 360-specific Tips and Tricks

Excellent Tips and Tricks video: https://youtu.be/hk4TN8xkSQM

## 3.1 Generative Designs that do not reconstruct correctly

Sometimes the design reconstruction process (i.e. result of **Create Design** from Outcome) does not complete successfully resulting in an error or fallback reconstruction that does not include an editable organic shape. Use of such models should be avoided.

The **Design History** of successfully reconstructed geometry appears as follows: 

If the organic shape (rounded blue cube) is not available for editing, try the following:

1. Look for locations material may have grown around open holes per Section 2.1.2. Make blind holes in **Preserves** if needed.
2. Turn through holes into blind holes, then extrude into through holes again after reconstruction.
3. Increase the **Synthesis Resolution** to maximum **Fine** and re-run the Generative **Study**.
4. Look for bad imported **Preserve** or **Obstacle** geometry, e.g. hollow parts rather than solid as indicated by section analysis or zero mass.
5. Eliminate small features such as thin **Preserves** and small holes in **Preserves**. Features smaller than ~1% of the maximum model dimension often cannot be represented by the voxel mesh. Constrain/load the surface, then put the hole in after model reconstruction.
6. Try reconstructing an earlier iteration in the design. 1-5 solutions back the design should not be much different but might reconstruct correctly.

## 3.2 Lightly loaded **Preserves** become separated/abandoned

For structures that hold components or react forces >10x different e.g. a 10 kg component and a 0.5kg component, sometimes the **Preserves** supporting the lightly loaded component become separated/abandoned. This can be identified by failure to **Converge** (e.g. only **Completed**) an error on the design solution, or a low number of iterations e.g. <20.

This can be corrected be either increasing the load on the separated **Preserves** with additional mass or force or manually connecting the preserves to more highly loaded preserves.

Alternatively, the following procedure can be followed:

1. Run an initial Generative Design study without the lightly loaded **Preserves** defined. This will create the main geometry supporting the highly loaded interfaces.
2. Choose a preferred Generative Design Output and save the design as a new part.
3. Begin a second Generative Design study using the result of the initial study generated in (2) & the light loaded structures as **Preserves.** This ensures the main load path is maintained and material will grow to accommodate the lightly loaded **Preserves**.
4. Delete all other **Preserves** used in the initial study.
5. Retain all **Obstacles** used in the initial study.
6. Keep **Mass Target** similar to the mass of the initial study result, but slightly higher to accommodate the new geometry that will be generated.
7. Validate the results with all loads as described in Section 7.

## 3.3 Large complex models fail to run or don't capture sufficient detail

The total number of voxels used to represent a design space is limited for computation reasons. Small or thin features require 2-3 voxels through the thickness for accurate results. For large and complex models, adding obstacles where material is unlikely to grow allows the use of smaller voxels in the relevant areas, increasing solution quality and speed. This can be done iteratively:

1. Run a generative design study
2. Add obstacles in regions where the results in (1) have no material (e.g. not in the load path)
3. Re-run generative study with new obstacles for improved result quality

## 3.4 Effect of Generative Design Resolution Setting

It is generally recommended to start with the default **Resolution** in **Study Settings**, which provides a good balance between solution speed and model detail. The Resolution setting determines the *total number* of voxels (cubic hex mesh elements) used to fill the design space. Resolution should be increased to (1) capture small details like thin preserves and small features on large models (2) accommodate complex models with many interfaces (3) increase the complexity of the design, e.g. more structural members, that can lead to more performant designs (4) help designs reconstruct properly as discussed in Section 3.1. However, increasing resolution increases run time and can lead to designs that are more complex to CNC machine as shown in Figure 3-1. Once a near final design is generated, it is recommended to try finer and/or coarser resolution to see if a more performant or manufacturable design can be achieved.

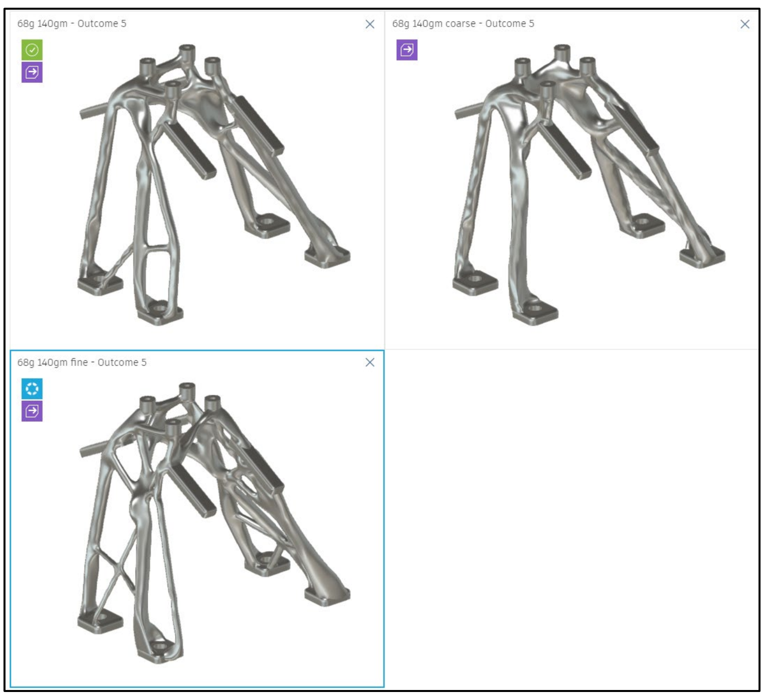



*Figure 3-1. Effect of the Resolution Setting. The **68g 140gm fine** solution in the lower left has a more complex structure, leading to better stiffness but more complex manufacturing. The **68g 140gm fine** solution in the upper right is less complex and less stiff, but may be easier to fabricate.*

# 4 Preparing design for FEA validation and fabrication

Before fabrication, Generatively Designed parts often need to be edited to ease manufacturing (e.g. reduce overhangs), clean up Generative Design artifacts (e.g. voxel print-through) and add features for secondary components (e.g. heater and corner cubes). FEA Validation (Section 7) should be performed after any CAD edits.

Protrusions and cuts to the solid geometry can be performed using any CAD software once the design is output as a STEP model however edits to the organic geometry must be done inside of Fusion 360's T-spline editing tool.

## 4.1 Deleting extraneous surfaces

During Generative Design optimization, **Obstacle**s are approximated by areas without voxels. However, after optimization, the precise **Obstacle** geometry is used to cut the resulting organic shape, often leaving behind extraneous surfaces. These may be deleted if there is no risk of interference to simplify fabrication and visually clean up the design as shown in Figure 4-1. Simply select all the surfaces and hit the delete key.


*Figure 4-1. Extraneous surfaces caused by obstacle cuts can be deleted to clean up the CAD geometry.*

## 4.2 Editing the organic shape (T-spline)

Direct editing of the organic shape can be used to simplify the design for manufacturing. Tips on T-spine editing for Generative Design can be found in this video guide: [Generative Design Master Class]() (starting at 32:30 minutes).

Of particular use is the **Cylindrify** command. During optimization, the structure is represented by voxels. Artifacts from the voxels during reconstruction can give parts a lumpy appearance. To simplify fabrication and improve the appearance of the part, nearly cylindrical structures can be made fully cylindrical as shown in Figure 4-2.

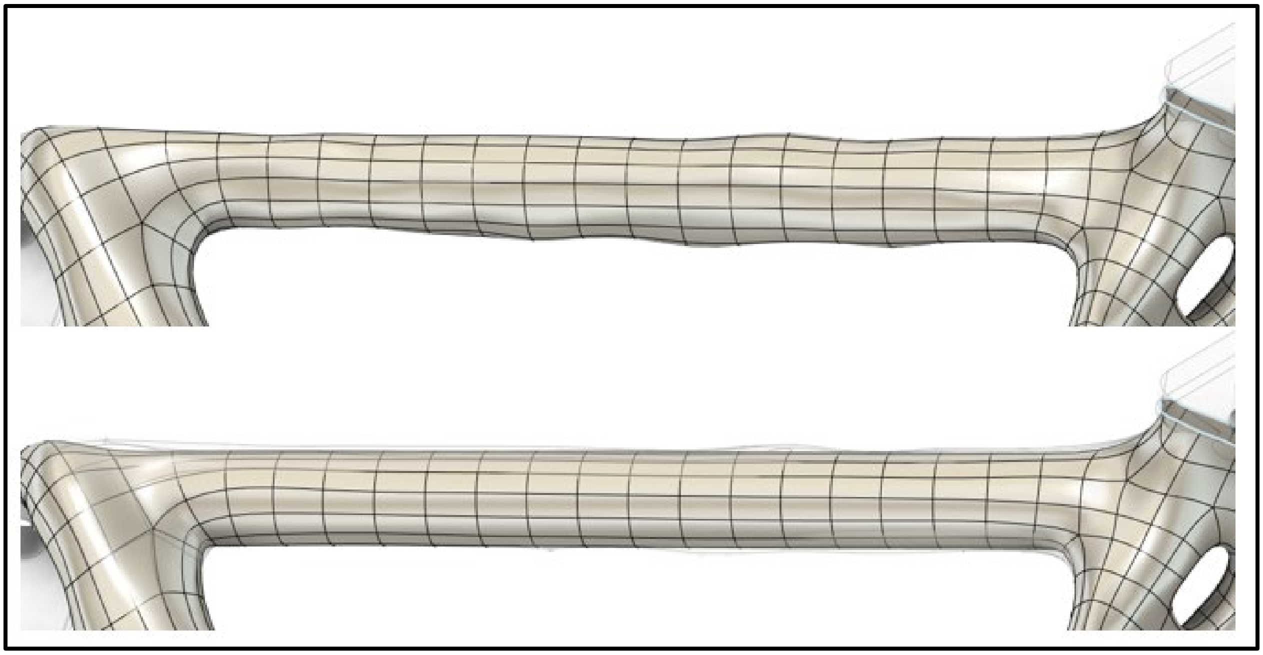


*Figure 4-2. Structural member before (top) and after (bottom)* ***Cylindrify*** *command.*

## 4.3 Adding additional/contingency mounting points

During Integration and Testing of structures, sometimes there in an unplanned need for additional mounting holes. This could be for wiring harness mounting, thermal blanket tie-downs, stray light baffling, or many other reasons. It is good practice to add additional mounting holes to Evolved Structures in strategic areas as shown in Figure 4-3. These holes can be modeled in Fusion 360 or project CAD (e.g. Creo or SolidWorks). Generally, tapped holes for small fasteners such as #6 or #4 are sufficient.

Typically, these mounting points don't carry much load and don't need to be considered in the Generative Design. If they do carry significant load, the validation FEM should be re-run confirming positive margins.

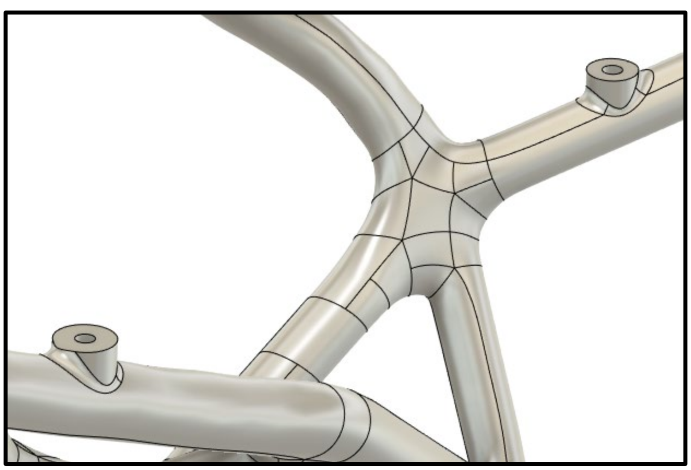


*Figure 4-3 Additional Mounting holes added to an Optical Bench for stray light baffles.*

## 4.4 Importing CAD into Creo

CAD models will generally be imported into the project's CAD software for integration into higher level assemblies, drawing creation, and formal release. In most CAD packages, STEP files from Fusion 360 import without issues. However, PTC Creo uses the Granite engine, from earlier generations of CAD when compound curved surfaces were uncommon, and can struggle to import STEP models correctly. If Creo does not import the STEP file as a solid, the following procedure can be used to fix the model. A copy of Autodesk Inventor is required to complete this task. This procedure also fixes issues with importing CAD into WindChill.

1. Export the part from Fusion 360 as Inventor (.ipt)
2. Open part in Autodesk Inventor and export file as granite (.g)
3. Open Creo and create a new part using the Goddard standard template.
4. File --> prepare --> model accuracy --> absolute 0.0004
5. Get data --> import --> select granite file
6. Regenerate part
7. Run model check and fix errors
8. Custom check-in. On second screen, select auto-resolve incomplete objects --> always ignore.

# 5 Detailed Finite Element Analysis (FEA) Validation

Before the design is fabricated, the design must be validated by detailed FEA. The voxel model used for optimization is coarse and will not capture localized stress peaks near interfaces. Detailed FEA can be performed by exporting the design to FEMAP or other pre-processors, but the Fusion 360 Simulation application is easy to use, based on the NASTRAN solver, and can greatly speed the validation process.

While the process below is generally sufficient for prototypes and engineering units, flight parts will be more extensively verified by FEA specialists, including dynamic analysis. This can be accelerated by exporting the NASTRAN data deck from Fusion 360 per Section TBD.

## 5.1 Static Analysis – Margin of Safety Verification

Loads, constraints, and materials will be automatically transferred from the Generative Design study, making static analysis very fast. See this video for a quick overview of the process: https://www.youtube.com/watch?v=Utz6CIbTtR4

### 5.1.1 Meshing

Default mesh settings (accessed by right clicking on the **Study**) are generally fine, but can be adjusted as desired, both globally and locally. Local mesh control on edges, faces, and bodies can be adjusted by selecting the geometry and right-clicking to access **Local Mesh Control**. FEA mesh sizing is beyond the scope of this guide. One default that should be changed is to uncheck **Create Curved Mesh Elements**, since this sometimes causes meshing failures and does not export the NASTRAN data deck well to FEMAP. Meshing issues are addressed in Section 5.3.

### 5.1.2 Margin of Safety Calculation

Once the static runs are complete, a Margin of Safety should be calculated for each load case based on accepted material Allowable Stress properties. For machined parts, these should be based on MMPDS values for the specific material, heat treatment, and specification that will be used for fabrication (e.g. AMS 4025/4027 for 6061-T651 Aluminum Plate). For AM parts, preliminary Allowable Stress can come from the part vendor but final values for flight parts should be developed per NASA-STD-6030 based on testing.

$$MS = \frac{\text{Allowable strength or stress}}{\text{Applied load or stress} * \text{Factors of safety}} - 1$$

Factors of Safety for Yield and Ultimate failure should be applied per NASA-STD-5001 (generally 1.4 for Ultimate and 1.25 for Yield). An example Margin Table is shown in Figure 5-1 below (see example spreadsheet). Any positive Margin is acceptable, but low margins from stress concentrations near bolted interfaces can be addressed per Section 7.3.

**Stress Margins**

Units are mks unless otherwise specified

| | |
|---|---|
| **Design** | *PSU protolabs4 0.8kg v4.step* |
| **FEA model** | *A19 STAR-X Vibe Analy* |
| **Material** | Aluminum 6061 - T651 |
| **Fty** | 241 |
| **Uts** | 290 |
| **Property source** | MMPDS |
| **FS yeild** | 1.25 |
| **FS ult** | 1.4 |

| Load Case | Max Von Mises (MPa) | Margin (yield) | Margin (Ultimate) |
|---|---|---|---|
| 12.6g x | 139.1 | 0.4 | 0.5 |
| 12.6g y | 90.9 | 1.1 | 1.3 |
| 12.6g z | 55.5 | 2.5 | 2.7 |

*Figure 5-1. Sample Stress Margin Table.*

### 5.1.3 Converging the Mesh

For areas of high stress, more accurate results can be obtained by converging the mesh size. Mesh convergence is achieved by iteratively reducing the element size until smaller elements do not affect the result of interest (e.g. stress, displacement, modes) within some threshold (e.g. 5%). This is automated in Fusion 360, while in other pre-processors such as FEMAP, it must be done manually. A single load case (usually the one with the lowest margin) must be selected to run a converged mesh study. In the study **Settings**, select **Adaptive Mesh Refinement**. The **Medium** or **High** setting usually gives good results with reasonable compute times and converges to 10% or 5% respectively.

## 5.2 Modal Analysis – 1st Mode Verification

Most structures have a minimum first mode requirement. This will be specified by the project or based on 542 Guidelines for Developing Preliminary Loads and Frequency Requirements (Alternate link) also shown in Appendix Table 5.

Create the modal analysis study by cloning the static analysis study (right click **Clone Study**, **Study Type Modal Frequencies**). **Number of Modes** 4 is generally sufficient. After completing the modal run, check the 1st mode is greater than the requirement.

## 5.3 Thermal Analysis

The same thermal modeling techniques used for complex traditional designs can also be applied to Evolved Structures. Thermal modeling of organic shapes can be challenging due to the increased computation cost of tetrahedral elements in thermal radiation analysis vs. structural analysis. The detail required in the thermal model depends on the environments the structure is exposed to and the thermal requirements for the structure. Techniques for thermal modeling of Evolved Structures include:

1. Model only conductance between interfaces. Aluminum and blanketed parts are usually conductance driven and radiation effects can often be ignored. Part conductance can be determined by applying a unit heat load to the structural FEM and measuring the temperature change at the interfaces.
2. Represent parts with thermal primitives (cylinders, bricks, etc.) that approximate the organic geometry. This is typically the responsibility of the thermal engineer, but the designer can help by creating curves and surfaces representing the design in CAD.
3. Represent the part with solid elements.
4. Use a detailed thermal model for conductance and a simplified model for radiation.

5. Use a SuperNetwork Thermal Desktop element, the thermal equivalent of NASTRAN Superelement.

## 5.4 Optical considerations

Compared to traditional designs, Generative Designs are often open and sparse. Stray light mitigation can be achieved with surface painting and blanketing with material appropriate to the instrument's wavelengths of interest. For instance, Black Kapton has been used for IR applications. Consult with coatings engineers for application specific needs.

Additional information regarding keeping optical pathways clear can be found in Section 2.6.1 - Light Path

## 5.5 NASTRAN Data Deck Export

Fusion 360 can export the NASTRAN data deck for editing and analysis other FEA pre/post-processors such as FEMAP or PATRAN and MSC or NX NASTRAN. This is not needed for Ansys since there is a more direct connection in the Fusion 360 interface.

Access to this NASTRAN data deck will greatly speed the work of downstream FEA including integration into assembly models for dynamics analysis.

You can create both high fidelity and lower fidelity models and even converge the mesh, something FEMAP cannot do easily.

Here is the process:

1. File -> View -> Show text commands
2. In the text commands pane, ensure the Txt radio button is selected
3. With the **study active**, type "SimFEACSExperimentalSolve.DebugFullReturn /on" (without the quotes) and press Enter
4. Send the study to cloud solve
5. The debug full return setting will persist with the study
6. When the solve has completed, go to the cache folder (*c:\users\<username>\AppData\local\Autodesk\Autodesk Fusion 360\<OxygenID>\W.login\Sim\<GUID>\SimResultsGUID\<GUID>*)
7. The file is called ***dbg-gewiz.nas***
8. You can also search *c:\users\<username>\AppData\local\Autodesk\Autodesk Fusion 360* for ****.nas*** or ***datemodified:10/3/2022 *.nas*** (obviously changing to your current date)

From FEMAP, just import the analysis model and select **Autodesk NASTRAN**. Before running the study in Fusion 360, turn off **Create Curved Mesh Elements** in the study settings. These advanced elements seem to give MSC and NX NASTRAN issues.

## 5.6 Common Problems and Solutions during FEA

### 5.6.1 Meshing Failures

The complex organic geometries created by Generative Design often fail to mesh, particularly near preserves and obstacles. First make sure **Create Curved Mesh Elements** is unchecked. Next look for small surfaces where indicated by the red error boxes.

#### *5.6.1.1 Self-Intersecting Meshes*

In Figure 5-2 we see an error of a self-intersecting mesh. While this error may appear to be in the rounded shoulder, the issue is that the organic extends inside the preserve with self-intersecting geometry. Self-intersecting might be anything from a tiny line segment that is attached to a vertex on one end and nothing else on the other. It may also be a sharp edge that, at the peak, curls back on itself making a 'ridge' that isn't attached to the solid.

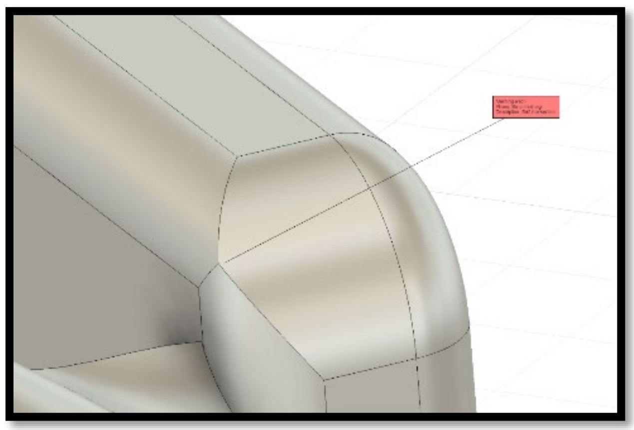


*Figure 5-2 - Example of error inside the solid model of the part*

A good hint to the problem is seen below:

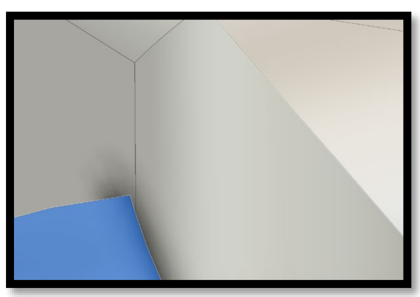


*Figure 5-3 - Sharp edge potential causing issue*

Leaving the "simulation" page and moving to the "design page will show our true culprit:

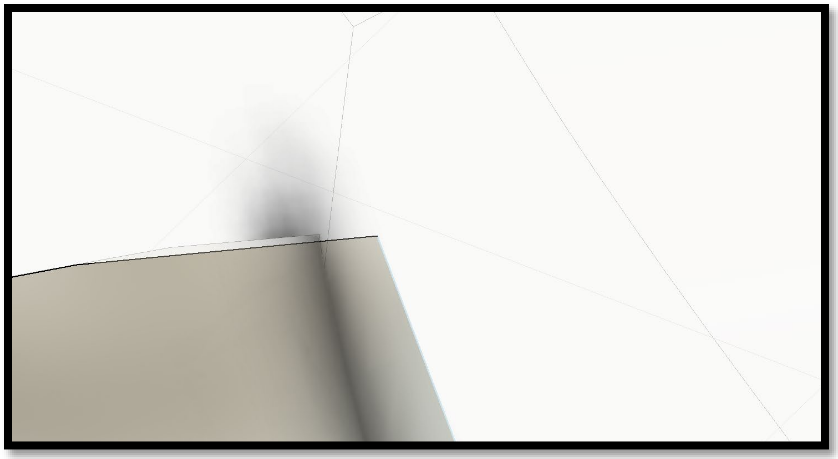


*Figure 5-4 - The white mesh in the background causing an issue with the mesh that extends inside the organic*

Use the Mesh edit tools by right clicking the purple box and select "edit" to modify the organic.

*Figure 5-5 - Timeline history with the purple "organic" that you can edit*

### 5.6.2 Disappearing sections after editing


If you edit an organic and afterward you click Finish Form:

But the organic you were working on disappears, you inadvertently exposed the 'inside' of the organic to open space (e.g. the organic is no longer water-tight). The solution is to look closely at what was edited and ensure that any form/organic terminates to or inside a surface.

### 5.6.3 FEA fails to run due to disconnected loads or constraints

If, during Generative Design, a **Preserve** surface used for a load or constraint is "grown over" with the organic shape the surface will be removed from the reconstructed design due to merging the organic with the **Preserves**. When the design is brought into **Simulation**, the load or constraint may become disconnected resulting in a failed **Simulation Study**. Sometimes no useful information is given about the failure.

To fix this error, either ensure the surface is not "grown over" using an obstacle, or choose a new surface to attach the load or constraint.

### 5.6.4 High Stress found in model

#### 5.6.4.1 Sliver surfaces (super thin) Elements

Sliver surfaces can lead to high aspect ratio elements and unrealistic stresses; edit the organic shape per section 4.2 to eliminate sliver surfaces.

#### 5.6.4.2 High Stress at Bolted Interfaces

Generative Designs usually have very low stress overall, leading to robust designs. However, unrealistically high stresses often occur near bolted interfaces, particularly constrained interfaces, due to the infinitely rigid NASTRAN RBE2 representation of the constraint. It is generally acceptable to use the maximum stress one element away from the constrained surface as show in Figure 5-6.

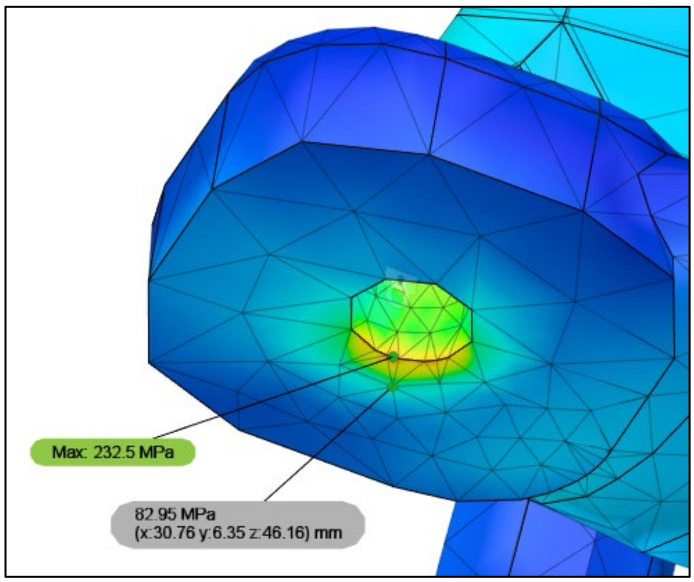



*Figure 5-6. Unrealistic high stress at constrained interface (232.5 MPs). Use maximum stress one element away (83 MPa) for Margin of Safety calculation.*

# 6 Tolerances and Inspection

Like any part, Evolved Structures must be inspected to ensure they will function properly before test verification and/or flight. Interfaces (e.g. bolted/bonded joints) can be inspected the same as traditionally designed parts. Tolerance should be as loose as possible for the part to meet functional requirements. Optical tolerances (e.g. 0.002” and below) are usually better achieved with adjustability (e.g. shims or threaded adjusters) to keep part costs reasonable. Modern CNC machines generally achieve +/-0.005” tolerances (0.014” true position on holes) without special CAM programming consideration. Flat sections on **Preserves**, such as bolted interfaces as describe in Section 2.1, are useful for creating tolerance/inspection Datum surfaces. When creating a drawing of the Evolved Structure, please consult the personnel that will inspect the part to ensure compatibility with available inspection tools and techniques.

For Evolves structures, the organic shapes can be spot checked with a CMM or Faro Arm to verify the profile tolerance. At a minimum, the location and size of each structural member should be checked. Since the organic shapes do not interface with other parts, they can generally have looser tolerances; a 0.020” profile tolerance is recommended. Mainly, inspection checks for gross errors, for instance if the wrong CAD model was used for fabrication.

Tolerances on the organic shapes could affect the modes of the part, and sensitivity analysis can be performed by editing the organic shapes in Fusion 360. In practice, as with traditionally designed parts, this is rarely done and it is often easier to verify modal performance by vibration or modal test. If desired, a full 3D scan of the Evolved structure can be performed, for instance using a Faro Arm with Blue Light scanner, to compare the entire part to the CAD model. The 3D scan can then be converted to a CAD model and re-analyzed to verify any expected change in performance. The details of this procedure depend on the software being used to scan and reconstruct the CAD, and are outside the scope of this document.

For AM parts, additional inspection for internal flaws may be required (e.g. CT or eddy current). Reference NASA-STD-6030.

# 7 Evolved Structures Representation in System Models

Structures are often represented in system-level structural and thermal models as simplified geometric primitives, such as plates and bars/beams to reduce model computation time. While the massive increase in computing power over the last decades has eased model size constraints, limitations remain. In general, Evolved Structures should be treated like traditionally designed geometrically complex parts; simplified to primitives when possible, and otherwise represented with the minimum number of solid/surface elements for the application.

## 7.1 Structural Dynamics

For structural dynamics models using in applications such as Coupled Loads Analysis (CLA), jitter, base drive sine, and random vibration reduced models should be used if the Evolved Structure requires too many solid elements, as determined by the project's Dynamics Subject Matter Expert (SME). The options for model reduction are:

1. Reduced solid element model
2. Simplification into beam elements
3. NASTRAN Super-element representation.

Evolved Structures often have long cylindrical sections as shown in Figure 20 (left). Like traditionally designed parts, a beam element FEM can be created and correlated with the detailed solid model, which is used for stress analysis.

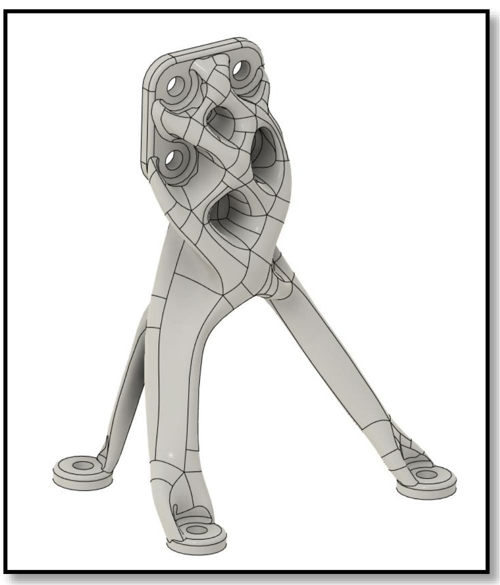


*Figure 7-1 - A15 Solution*

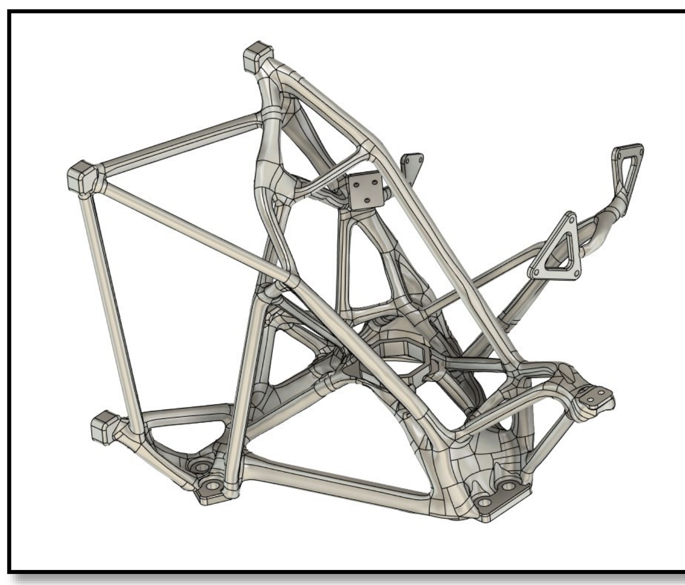


*Figure 7-2- A18 Solution*

For more compact parts that are not easily represented by beam elements (see Figure 7-1 and 7-2) a reduced solid (tet10) element model may be used and correlated against the detailed FEM. A good rule-of-thumb is to have at least 2 elements through the thickness of structural sections.

For complex parts, or even entire sub-systems, NASTRAN Super-element representation may be used. This video provides a good primer on Super-elements. Super-elements represent the stiffness and dynamic behavior of the structure with a matrix; physical degrees of freedom are only included at the interfaces/boundaries. Responses are the interfaces may then be extracted and applied to the detailed FEM if required for detailed stress analysis.

## 7.2 Structural Thermal Optical Performance (STOP) Analysis

For high-precision optical systems, STOP analysis is used to predict the effect of thermal distortion on optical performance. STOP requires a FEM, thermal model, and optical performance model.

Evolved Structures may be represented in STOP FEMs via (1) simplification into beam elements (2) reduced solid element model as described in Section 7.1. STOP FEM run time is less sensitive to large numbers of elements compared to dynamic FEMs. Unless there are redundant load paths, thermal distortion in not sensitive to part stiffness since thermal distortion is proportional to the temperature change * length.

Thermal modeling is addresses in Section 0, and optical modeling is independent of the structures.

# 8 Appendix

## 8.1 Clearance Hole **Preserves** Dimensions

| Bolt Size (Imperial) | Tapped Hole Diameter (inches) | Normal Clearance Hole Diameter (inches) | Minimum Outside Dimensions (inches) | Preserve Fillet (inches) | Washer Diameter (inches) (Approx. NAS620) | Recommended Directly-Threaded Depth (in) | Recommended Directly-Threaded Part Thickness (in) | Obstacle filet (in) |
|---|---|---|---|---|---|---|---|---|
| #0 | 0.047 | 0.076 | 0.228 | 0.0304 | 0.10 | 0.0705 | 0.0940 | 0.032 |
| #1 | 0.06 | 0.089 | 0.267 | 0.0356 | 0.15 | 0.09 | 0.1200 | 0.032 |
| #2 | 0.07 | 0.102 | 0.306 | 0.0408 | 0.15 | 0.105 | 0.1400 | 0.032 |
| #3 | 0.079 | 0.116 | 0.348 | 0.0464 | 0.18 | 0.1185 | 0.1580 | 0.032 |
| #4 | 0.089 | 0.128 | 0.348 | 0.0464 | 0.21 | 0.1335 | 0.1780 | 0.032 |
| #5 | 0.102 | 0.156 | 0.384 | 0.0512 | 0.24 | 0.153 | 0.2040 | 0.032 |
| #6 | 0.107 | 0.17 | 0.51 | 0.068 | 0.27 | 0.1605 | 0.2140 | 0.032 |
| #8 | 0.136 | 0.196 | 0.588 | 0.0784 | 0.30 | 0.204 | 0.2720 | 0.032 |
| #10 | 0.15 | 0.221 | 0.663 | 0.0884 | 0.35 | 0.225 | 0.3000 | 0.032 |
| 1/4" | 0.201 | 0.281 | 0.843 | 0.1124 | 0.47 | 0.3015 | 0.4020 | 0.016 |
| 5/16" | 0.257 | 0.344 | 1.032 | 0.1376 | 0.56 | 0.3855 | 0.5140 | 0.016 |
| 3/8" | 0.313 | 0.406 | 1.218 | 0.1624 | 0.56 | 0.4695 | 0.6260 | 0.016 |
| 7/16" | 0.368 | 0.469 | 1.407 | 0.1876 | 1.13 | 0.552 | 0.7360 | 0.016 |
| 1/2" | 0.422 | 0.562 | 1.686 | 0.2248 | 0.81 | 0.633 | 0.8440 | 0.016 |
| 5/8" | 0.531 | 0.688 | 2.064 | 0.2752 | 1.25 | 0.7965 | 1.0620 | 0.032 |
| 3/4" | 0.656 | 0.812 | 2.436 | 0.3248 | 0.38 | 0.984 | 1.3120 | 0.032 |
| 7/8" | 0.766 | 0.938 | 2.814 | 0.3752 | 0.63 | 1.149 | 1.5320 | 0.032 |
| 1" | 0.875 | 1.094 | 3.282 | 0.4376 | 2.00 | 1.3125 | 1.7500 | 0.032 |

## 8.2 Clearance Hole for Inch Fasteners

*Table A8-1. Clearance Holes for English Fasteners (reference only). Source: https://www.eng-tips.com/viewthread.cfm?qid=483545*

**Table 1a. Clearance Holes for Inch Fasteners** *ASME B18.2.8-1999, R2005*

| Nominal Screw Size | Normal | | | Close | | | Loose | | |
|---|---|---|---|---|---|---|---|---|---|
| | Nominal Drill Size | Hole Diameter | | Nominal Drill Size | Hole Diameter | | Nominal Drill Size | Hole Diameter | |
| | | Min. | Max. | | Min. | Max. | | Min. | Max. |
| #0 | #48 | 0.076 | 0.082 | #51 | 0.067 | 0.071 | 3/32 | 0.094 | 0.104 |
| #1 | #43 | 0.089 | 0.095 | #46 | 0.081 | 0.085 | #37 | 0.104 | 0.114 |
| #2 | #38 | 0.102 | 0.108 | 3/32 | 0.094 | 0.098 | #32 | 0.116 | 0.126 |
| #3 | #32 | 0.116 | 0.122 | #36 | 0.106 | 0.110 | #30 | 0.128 | 0.140 |
| #4 | #30 | 0.128 | 0.135 | #31 | 0.120 | 0.124 | #27 | 0.144 | 0.156 |
| #5 | 5/32 | 0.156 | 0.163 | 9/64 | 0.141 | 0.146 | 11/64 | 0.172 | 0.184 |
| #6 | #18 | 0.170 | 0.177 | #23 | 0.154 | 0.159 | #13 | 0.185 | 0.197 |
| #8 | #9 | 0.196 | 0.203 | #15 | 0.180 | 0.185 | #3 | 0.213 | 0.225 |
| #10 | #2 | 0.221 | 0.228 | #5 | 0.206 | 0.211 | B | 0.238 | 0.250 |
| 1/4 | 9/32 | 0.281 | 0.290 | 17/64 | 0.266 | 0.272 | 19/64 | 0.297 | 0.311 |
| 5/16 | 11/32 | 0.344 | 0.354 | 21/64 | 0.328 | 0.334 | 23/64 | 0.359 | 0.373 |
| 3/8 | 13/32 | 0.406 | 0.416 | 25/64 | 0.391 | 0.397 | 27/64 | 0.422 | 0.438 |
| 7/16 | 15/32 | 0.469 | 0.479 | 29/64 | 0.453 | 0.460 | 31/64 | 0.484 | 0.500 |
| 1/2 | 9/16 | 0.562 | 0.572 | 17/32 | 0.531 | 0.538 | 39/64 | 0.609 | 0.625 |
| 5/8 | 11/16 | 0.688 | 0.698 | 21/32 | 0.656 | 0.663 | 47/64 | 0.734 | 0.754 |
| 3/4 | 13/16 | 0.812 | 0.824 | 25/32 | 0.781 | 0.789 | 1 29/32 | 0.906 | 0.926 |
| 7/8 | 15/16 | 0.938 | 0.950 | 29/32 | 0.906 | 0.914 | 1 1/32 | 1.031 | 1.051 |
| 1 | 1 3/32 | 1.094 | 1.106 | 1 1/32 | 1.031 | 1.039 | 1 5/32 | 1.156 | 1.181 |
| 1 1/8 | 1 7/32 | 1.219 | 1.235 | 1 5/32 | 1.156 | 1.164 | 1 5/16 | 1.312 | 1.337 |
| 1 1/4 | 1 11/32 | 1.344 | 1.360 | 1 9/32 | 1.281 | 1.291 | 1 7/16 | 1.438 | 1.463 |
| 1 3/8 | 1 1/2 | 1.500 | 1.516 | 1 7/16 | 1.438 | 1.448 | 1 39/64 | 1.609 | 1.634 |
| 1 1/2 | 1 5/8 | 1.625 | 1.641 | 1 9/16 | 1.562 | 1.572 | 1 47/64 | 1.734 | 1.759 |

**Table 1b. Inch Clearance Hole Allowances**

| Nominal Screw Size | Fit Classes | | | Nominal Screw Size | Fit Classes | | |
|---|---|---|---|---|---|---|---|
| | Normal | Close | Loose | | Normal | Close | Loose |
| #0 – #4 | 1/64 | 0.008 | 1/32 | 1 | 3/32 | 1/32 | 5/32 |
| #5 – 7/16 | 1/32 | 1/64 | 3/64 | 1 1/8, 1 1/4 | 3/32 | 1/32 | 3/16 |
| 1/2, 5/8 | 1/16 | 1/32 | 7/64 | 1 3/8, 1 1/2 | 1/8 | 1/16 | 15/64 |
| 3/4, 7/8 | 1/16 | 1/32 | 5/32 | ... | ... | ... | ... |

Dimensions are in inches.

## 8.3 Blind and Threaded Hole Dimensions

*Table A8-2. Drill sizes for English fasteners. Source [https://en.wikipedia.org/wiki/Unified_Thread_Standard](https://en.wikipedia.org/wiki/Unified_Thread_Standard)*

**Unified Screw Threads — UNC, UNF and UNEF[2]:1816**

| Major diameter $D_{maj}$ (inch, mm) | | | Thread density (*d*, threads per inch) and thread pitch (*p*) | | | | | | | | | Preferred cutting tap drill size | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | Coarse (UNC) | | | Fine (UNF) | | | Extra fine (UNEF) | | | | | | | | |
| | | | *d* (TPI) | *p* (inch, mm) | | *d* (TPI) | *p* (inch, mm) | | *d* (TPI) | *p* (inch, mm) | | Coarse | | Fine | | Extra fine | |
| #0 | 0.0600 | 1.5240 | None | | | 80 | 0.012500 | 0.3175 | None | | | | | $\frac{3}{64}$ in | .047 | | |
| #1 | 0.0730 | 1.8542 | 64 | 0.015625 | 0.3969 | 72 | 0.013888 | 0.3528 | None | | | #53 | .060 | #53 | .060 | | |
| #2 | 0.0860 | 2.1844 | 56 | 0.017857 | 0.4536 | 64 | 0.015625 | 0.3969 | None | | | #50 | .070 | #50 | .070 | | |
| #3 | 0.0990 | 2.5146 | 48 | 0.020833 | 0.5292 | 56 | 0.017857 | 0.4536 | None | | | #47 | .079 | #45 | .082 | | |
| #4 | 0.1120 | 2.8448 | 40 | 0.025000 | 0.6350 | 48 | 0.020833 | 0.5292 | None | | | #43 | .089 | #42 | .094 | | |
| #5 | 0.1250 | 3.1750 | 40 | 0.025000 | 0.6350 | 44 | 0.022727 | 0.5773 | None | | | #38 | .102 | #37 | .104 | | |
| #6 | 0.1380 | 3.5052 | 32 | 0.031250 | 0.7938 | 40 | 0.025000 | 0.6350 | None | | | #36 | .107 | #33 | .113 | | |
| #8 | 0.1640 | 4.1656 | 32 | 0.031250 | 0.7938 | 36 | 0.027778 | 0.7056 | None | | | #29 | .136 | #29 | .136 | | |
| #10 | 0.1900 | 4.8260 | 24 | 0.041667 | 1.0583 | 32 | 0.031250 | 0.7938 | None | | | #25 | .150 | #21 | .159 | | |
| #12 | 0.2160 | 5.4864 | 24 | 0.041667 | 1.0583 | 28 | 0.035714 | 0.9071 | 32 | 0.031250 | 0.7938 | #16 | .177 | #14 | .182 | $\frac{3}{16}$ in | .188 |
| $\frac{1}{4}$" | 0.2500 | 6.3500 | 20 | 0.050000 | 1.2700 | 28 | 0.035714 | 0.9071 | 32 | 0.031250 | 0.7938 | #7 | .201 | #3 | .213 | $\frac{7}{32}$ in | .219 |
| $\frac{5}{16}$" | 0.3125 | 7.9375 | 18 | 0.055556 | 1.4111 | 24 | 0.041667 | 1.0583 | 32 | 0.031250 | 0.7938 | F | .257 | I | .272 | $\frac{9}{32}$ in | .281 |
| $\frac{3}{8}$" | 0.3750 | 9.5250 | 16 | 0.062500 | 1.5875 | 24 | 0.041667 | 1.0583 | 32 | 0.031250 | 0.7938 | $\frac{5}{16}$ in | .313 | Q | .332 | $\frac{11}{32}$ in | .344 |
| $\frac{7}{16}$" | 0.4375 | 11.1125 | 14 | 0.071428 | 1.8143 | 20 | 0.050000 | 1.2700 | 28 | 0.035714 | 0.9071 | U | .368 | $\frac{25}{64}$ in | .391 | Y | .404 |
| $\frac{1}{2}$" | 0.5000 | 12.7000 | 13 | 0.076923 | 1.9538 | 20 | 0.050000 | 1.2700 | 28 | 0.035714 | 0.9071 | $\frac{27}{64}$ in | .422 | $\frac{29}{64}$ in | .453 | $\frac{15}{32}$ in | .469 |
| $\frac{9}{16}$" | 0.5625 | 14.2875 | 12 | 0.083333 | 2.1167 | 18 | 0.055556 | 1.4111 | 24 | 0.041667 | 1.0583 | $\frac{31}{64}$ in | .484 | $\frac{1}{2}$ in | .500 | $\frac{33}{64}$ in | .516 |
| $\frac{5}{8}$" | 0.6250 | 15.8750 | 11 | 0.090909 | 2.3091 | 18 | 0.055556 | 1.4111 | 24 | 0.041667 | 1.0583 | $\frac{17}{32}$ in | .531 | $\frac{9}{16}$ in | .563 | $\frac{37}{64}$ in | .578 |
| $\frac{3}{4}$" | 0.7500 | 19.0500 | 10 | 0.100000 | 2.5400 | 16 | 0.062500 | 1.5875 | 20 | 0.050000 | 1.2700 | $\frac{21}{32}$ in | .656 | $\frac{11}{16}$ in | .688 | $\frac{45}{64}$ in | .703 |
| $\frac{7}{8}$" | 0.8750 | 22.2250 | 9 | 0.111111 | 2.8222 | 14 | 0.071428 | 1.8143 | 20 | 0.050000 | 1.2700 | $\frac{49}{64}$ in | .766 | $\frac{51}{64}$ in | .797 | $\frac{53}{64}$ in | .828 |
| 1" | 1.0000 | 25.4000 | 8 | 0.125000 | 3.1750 | 12[a] | 0.083333 | 2.1167 | 20 | 0.050000 | 1.2700 | $\frac{7}{8}$ in | .875 | $\frac{59}{64}$ in | .922 | $\frac{61}{64}$ in | .953 |

## 8.4 Washer **Obstacles**

*Table A8-3. NAS620 washer sizes. Source: McMaster-Carr.*

| For Screw Size | ID | OD | Thick. | Hardness | Specifications Met | Pkg. Qty. | | Pkg. |
|---|---|---|---|---|---|---|---|---|
| **Passivated 18-8 Stainless Steel** | | | | | | | | |
| **NAS620** | | | | | | | | |
| No. 0 | 0.063" | 0.099" | 0.015"-0.018" | Rockwell B70 | NAS620C0 | 250 | 90945A700 | $27.29 |
| No. 2 | 0.089" | 0.149" | 0.015"-0.018" | Rockwell B70 | NAS620C2 | 500 | 90945A705 | 27.08 |
| No. 3 | 0.102" | 0.180" | 0.015"-0.018" | Rockwell B70 | NAS620C3L | 500 | 90945A707 | 34.58 |
| No. 3 | 0.102" | 0.180" | 0.030"-0.034" | Rockwell B70 | NAS620C3 | 500 | 90945A708 | 34.17 |
| No. 4 | 0.115" | 0.209" | 0.015"-0.018" | Rockwell B70 | NAS620C4L | 500 | 90945A710 | 29.17 |
| No. 4 | 0.115" | 0.209" | 0.030"-0.034" | Rockwell B70 | NAS620C4 | 500 | 90945A711 | 32.50 |
| No. 5 | 0.128" | 0.238" | 0.015"-0.018" | Rockwell B70 | NAS620C5L | 500 | 90945A712 | 40.42 |
| No. 5 | 0.128" | 0.238" | 0.030"-0.034" | Rockwell B70 | NAS620C5 | 250 | 90945A713 | 25.62 |
| No. 6 | 0.143" | 0.267" | 0.015"-0.018" | Rockwell B70 | NAS620C6L | 250 | 90945A715 | 18.33 |
| No. 6 | 0.143" | 0.267" | 0.030"-0.034" | Rockwell B70 | NAS620C6 | 250 | 90945A716 | 20.21 |
| No. 8 | 0.169" | 0.304" | 0.015"-0.018" | Rockwell B70 | NAS620C8L | 250 | 90945A725 | 19.17 |
| No. 8 | 0.169" | 0.304" | 0.030"-0.034" | Rockwell B70 | NAS620C8 | 250 | 90945A726 | 22.71 |
| No. 10 | 0.195" | 0.354" | 0.030"-0.034" | Rockwell B70 | NAS620C10L | 250 | 90945A740 | 27.08 |
| No. 10 | 0.195" | 0.354" | 0.060"-0.066" | Rockwell B70 | NAS620C10 | 100 | 90945A741 | 21.25 |
| 1/4" | 0.255" | 0.468" | 0.030"-0.034" | Rockwell B70 | NAS620C416L | 100 | 90945A760 | 18.17 |
| 1/4" | 0.255" | 0.468" | 0.060"-0.066" | Rockwell B70 | NAS620C416 | 100 | 90945A761 | 31.58 |

## 8.5 GEVS-based Loads and Modal requirement assumptions

*Table 8-4. Mass Acceleration Curve (MAC) loads per 542 standards. Linear interpolation can be used between masses.*

| Hardware Mass (kg) | Limit Load (G, any direction) | Acceleration $^{m}/_{s^2}$ |
|---|---|---|
| **1 or less** | 68 | 667.08 |
| **5** | 49 | 480.69 |
| **10** | 40 | 392.40 |
| **20** | 31 | 304.11 |
| **30** | 27 | 264.87 |
| **40** | 24 | 235.44 |
| **50 or greater** | 22 | 215.82 |

*Table A8-5. Generic frequency requirements per 542 standards.*

| Structure Type | Requirement | Rationale |
|---|---|---|
| Deployables and Large Instrument/Subsystems (> 500 lbs.) | > 35 Hz | Dynamic input to the payload for most ELV's occurs below 35 Hz. Keeping hardware above 35 Hz limits the amount of dynamic coupling with the low-frequency launch environment. |
| Electronics Boxes and Small Components (< 50 lbs.) | > 100 Hz | Most ELV's define their sine specs and the low-frequency launch environment to extend out to 100 Hz. By keeping hardware above 100 Hz, there are no issues about qualification for the launch environment in the 50 to 100 Hz range. The item can also be treated as a lumped mass in the coupled loads analysis. |
| Other Components/Subsystems | > 50 Hz | Most ELV coupled loads analysis cuts off at 50 Hz. If a component is above 50 Hz, then a modal survey and a test verified model is not required. A detailed FEM is still required for the coupled loads analysis which reflects all significant modes up to at least 75 Hz. |