Structural Analysis Using Computer Based Methods

Matthew R Dietz[1]

NASA NE-M1, University of Florida, Gainesville, FL, 32608

The stiffness of a flex hose that will be used in the umbilical arms of the Space Launch Systems mobile launcher needed to be determined in order to properly qualify ground umbilical plate behavior during vehicle separation post T-0. This data is also necessary to properly size and design the motors used to retract the umbilical arms. Therefore an experiment was created to determine the stiffness of the hose. Before the test apparatus for the experiment could be built, the structure had to be analyzed to ensure it would not fail under given loading conditions. The design model was imported into the analysis software and optimized to decrease runtime while still providing accurate results and allow for seamless meshing. Areas exceeding the allowable stresses in the structure were located and modified before submitting the design for fabrication. In addition, a mock up of a deep space habitat and the support frame was designed and needed to be analyzed for structural integrity under different loading conditions. The load cases were provided by the customer and were applied to the structure after optimizing the geometry. Once again, weak points in the structure were located and recommended design changes were made to the customer and the process was repeated until the load conditions were met without exceeding the allowable stresses. After the stresses met the required factors of safety the designs were released for fabrication.

Nomenclature

A	=	cross sectional area
σ	=	normal stress
τ	=	shear stress
F_x	=	x component of the resultant force
F_y	=	y component of the resultant force
R_x	=	reaction force in the x direction
R_y	=	reaction force in the y direction
ipart	=	idealized part
FEM	=	finite element model
sim	=	simulation model

I. Introduction

Before any structure or test apparatus can be built it must first be analyzed to ensure the appropriate NASA Ground Support Equipment (GSE) or Flight Hardware standards are met. Computer models are simplified and prepared for analysis before applying the given loading conditions that the structure is expected to see. If the model does not meet the specified standards, the structure must be redesigned or modified before proceeding to the next stage in the design process. It is the structures analyst's responsibility to verify that the structures and components analyzed meet all relevant requirements and if there are areas of concern, present the customer with these issues or concerns along with possible fixes/solutions so that they may be resolved.

II. Creating Structural Analysis Model

This section will outline the process of going from the three dimensional design model to a simplified model that decreased solution runtimes and help facilitate meshing of the structure. Without these steps, models can take hours to run and provide inaccurate results due to problems with the mesh.

A. Simplifying Geometry - The ipart

Initially, the imported design has unnecessary geometry and overly complicated members. This includes, some fillets, holes, non-structural members, and unnecessary three dimensional pieces. To prevent long run times, these unneeded geometries are removed or simplified if possible. Long run times slow the work flow and can cause

---

[1] Structural Analysis Intern, NE-M1, Kennedy Space Center, University of Florida.

projects to miss deadlines. Long run times also make trouble shooting more difficult. For example, if a model takes a 3 hours to run and a error that has a simple solution occurs at the end of the run then the time spent running was wasted. Whereas, if the model only took 10 minutes to run then the error would be fixed much quicker and work flow left uninterrupted. In addition to decreasing runtime, simplifying the model also allows for better meshing which will lead to more accurate results.

Depending on what is needed from the results, the model can be modeled using representative three dimensional, two dimensional, or one dimensional elements. Trusses, for example, can be converted to one dimensional elements using lines in place of the 3D members if the axial stress and general response to the loads is needed. But if greater detail is necessary in specific areas then a shell, or 2 dimensional, model is more appropriate.

In the case of the flex hose test structure, the deep space habitat node, and accompanying support structure, 3D elements were converted to 2D elements by adding a plane between the two surfaces of the element piece. However, some members were left as 3D elements because converting them created problems with the geometry as a whole. In short, most detailed models will consist of 1D, 2D, and 3D elements.
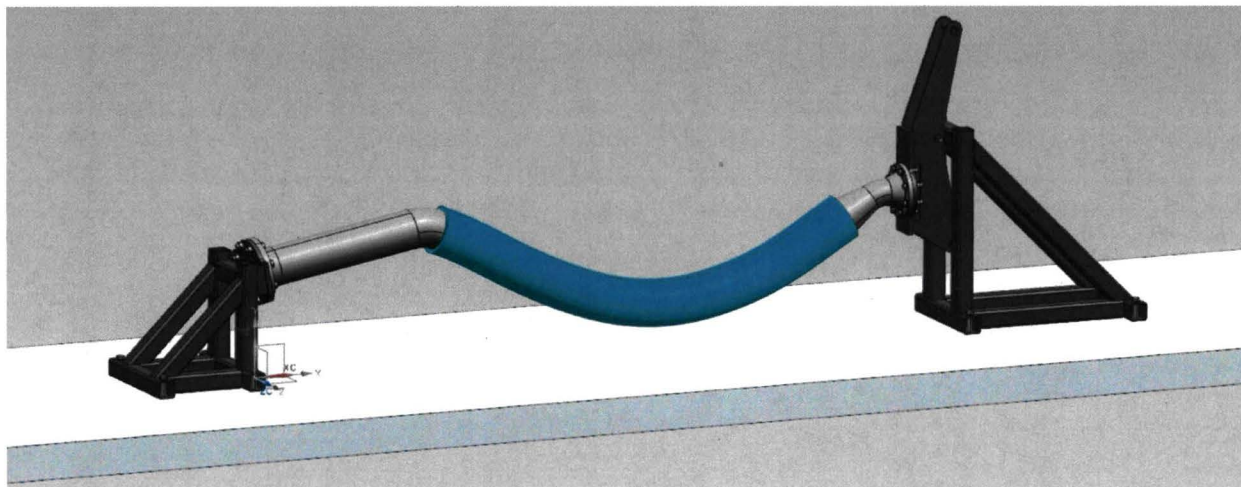

Figure 1. Imported Geometry of the Flex Hose test structure.

Figure 2. Process of simplifying the geometry of the structure to decrease run time.

Figures 1 and 2 above show the process of simplifying the geometry of the flex hose test structure, from the designer's model to the analyst's model. The structure was broken up into two pieces and unnecessary parts were removed such as the hose, couplings, bolts, etc. The fillets on the beams were then removed and shells were created along the centerline of the plates and beams. Doing this creates gaps between the shells and they must be extended before continuing.

B. Creating the Mesh - The FEM

Once the geometry has been simplified, the pieces have to be joined together and just because they are touching does not mean contact between the pieces is recognized. There are several ways to do this. The first way is to try using the stitch command. After doing this, the geometry must be inspected for issues caused by the stitching. If this does not work there are other methods but those are used later in the modeling process.

A finite element model (FEM) is created by meshing each piece. A mesh creates interconnected nodes on the structure where values such as displacement, stress, and reaction forces, are calculated. Parts with different thickness, material, and other material properties must be meshed separately because these properties are defined on a material card that is associated with each mesh. There are several different types of meshes that are used in different cases, 0D, 1D, 2D, 3D, and there are different ways of meshing each. Bolted connections, when needed are modeled using 1D beams connected to each node along the edge of the hole.
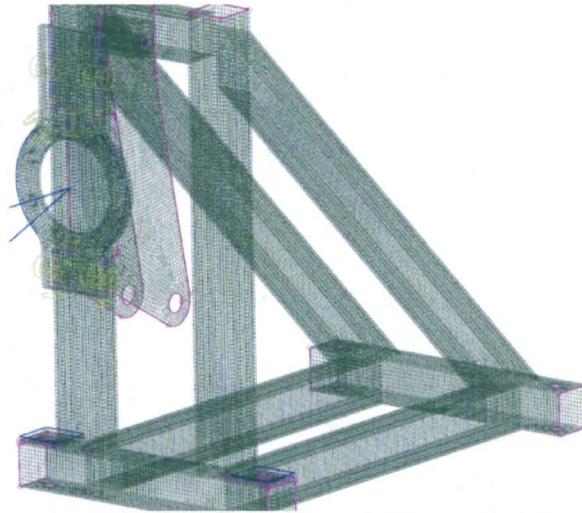

Figure 3. The mesh is difficult to see due to the small element size. Pink lines indicate free edges that are not connected to anything.

C. Setting Parameters - The sim

After the FEM is created the next step is setting the parameters and constraints for the model. This includes applying forces caused by loading conditions, gravity, accelerations etc. In addition to applying forces, boundary conditions are also created to keep the structure from moving and to simulate attachment points and degrees of freedom of the structure. Together, the boundary conditions and the applied forces work to simulate the environment the structure will be subjected to.
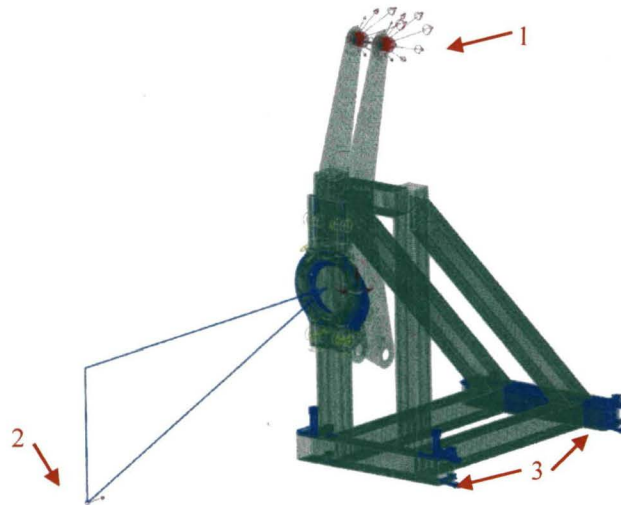
Figure 4. The red arrows represent applied forces and the blue hatch marks are boundary conditions.

Figure 4 shows the simulation model (sim) for the flex hose test structure. The area indicated by (1) is a bearing load simulating a load being applied to a pin; as a result, the force is distributed along the edge of the holes as shown. Arrow (2) shows a load that simulates an infinitely stiff hose which is considered the worst case scenario. This force was found by doing a hand calculation to find a force that creates an equal and opposite moment at the point of rotation. Finally (3) denotes the areas where boundary conditions representing the bolts that hold the structure to the ground are applied

III. Interpreting the Results

After the analysis model has been run, it is important to interpret the results to ensure that the model was accurately modeled and that the results are plausible under the given loading condition. In the case of the flex hose test, the movement of the structure was checked to make sure that the reactions at the fixture points were as expected given the applied loads and boundary conditions and that the stresses in the structure were appropriate.
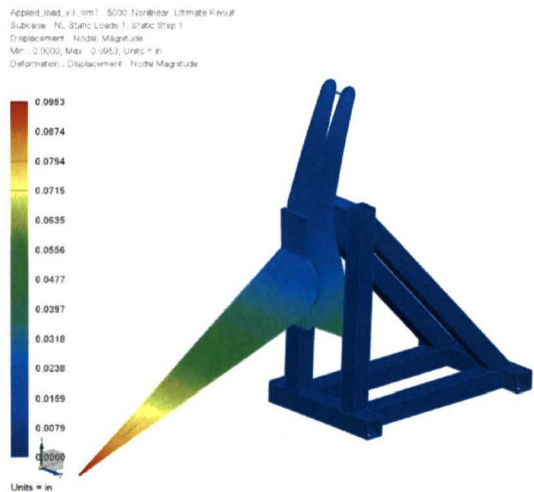

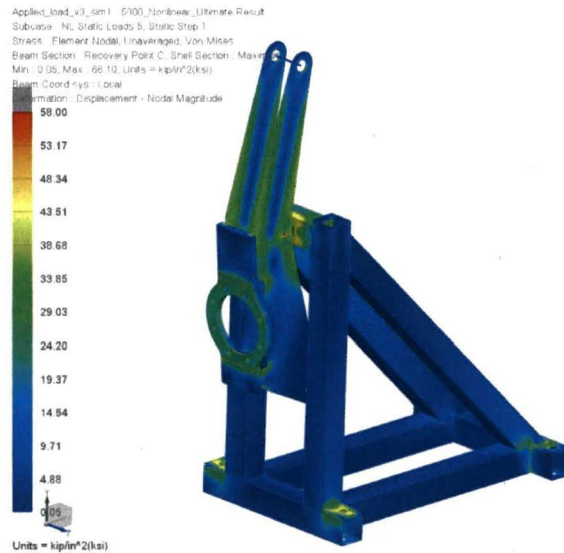**Figure 5. Color map showing the displacement of the structure.**

Figure 6. Color map of the stresses experienced by the structure.

A. Factor of Safety

Specific factors of safety were used to determine the allowable stress in the structure. A factor of safety of 2 was used on yield stresses and a factor of safety of 3 was used on ultimate stresses, in accordance with KSC-STD-5005c GSE standards. To determine the allowable stresses, the following equation was used:

$$\sigma_{alloable} = \frac{\sigma}{Factor\ of\ safety}$$

Using this equation, the lowest allowable stress is determined and is used as the driving stress factor for the structure.
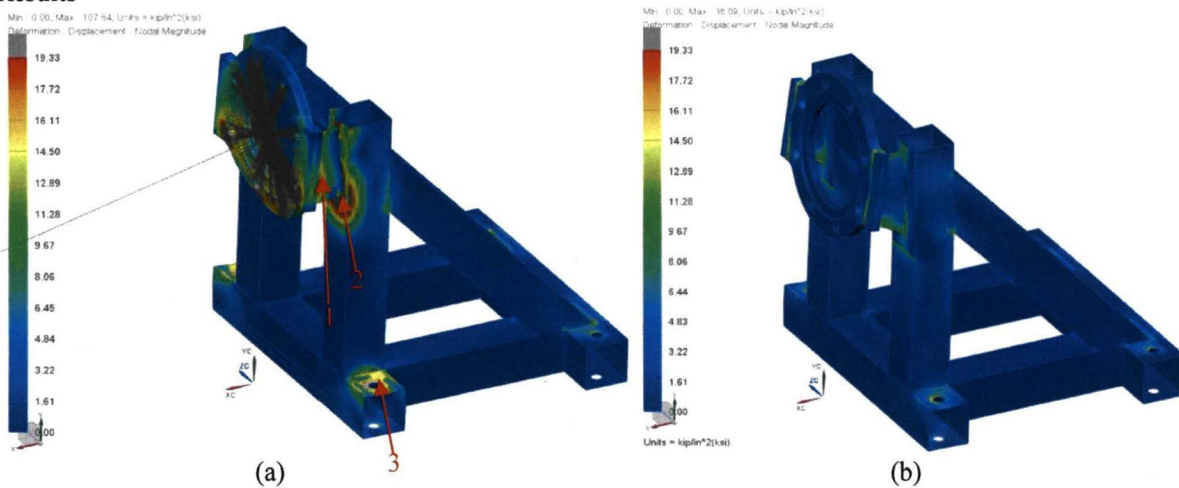
B. Results

(a)   (b)

Figure 7. Stress plots before and after modifications were made to meet allowable stresses

Figure 7a shows the structure before modifications were made. There areas in gray represent areas where the stress in the structure exceed the allowable stress of 19.3 ksi. Modifications were made to the structure until all stresses were below the 19.3 ksi allowable. The connecting plate indicated as (1) was increased from 0.375" to 0.5" and a gusset was added to better distribute the loads. Also, a 0.5" plate was added to the area behind the connecting plate to distribute the load caused by the hose across a larger area of the beam, shown as (2). Finally, A 0.5" plate was added to the front feet, (3), to act as an over sized washer and resist the prying motion caused by the hose. Figure 7b shows the results of these changes.

IV. Conclusion

Without structural analysis, structures could be unstable and potentially dangerous. Analysis shows areas of concern allowing the item to be redesigned before it is produced therefore saving time and money. In addition, it also creates a safer environment for the customer and prevents injury due to failure of the structure. This design and analysis process can be repeated many times before a final design is determined.

Appendix A

Modeling Process of the Deep Space Habitat and support frame
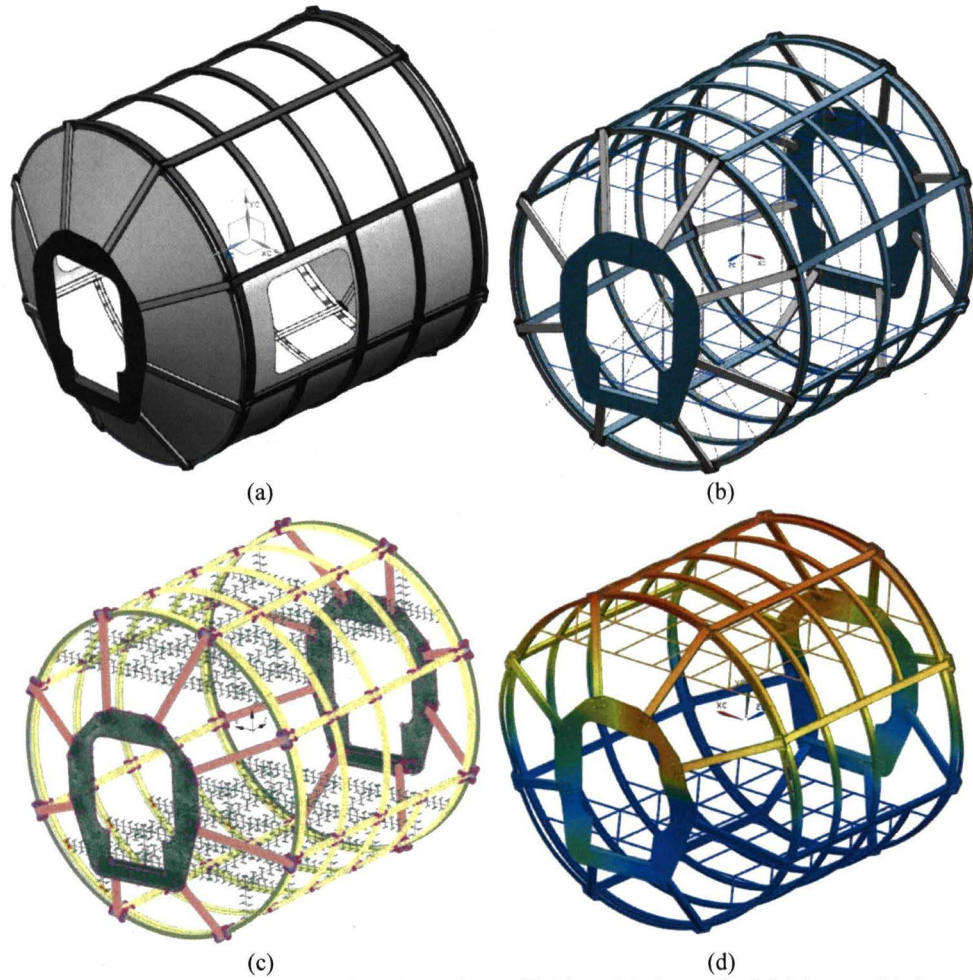

(a)

(b)

(c)

(d)

Figure 8. Analysis process for a mock up of the Deep Space Habitat. (a) shows the initial part, (b) shows the ipart, (c) shows the FEM, and (d) shows the displacement results.
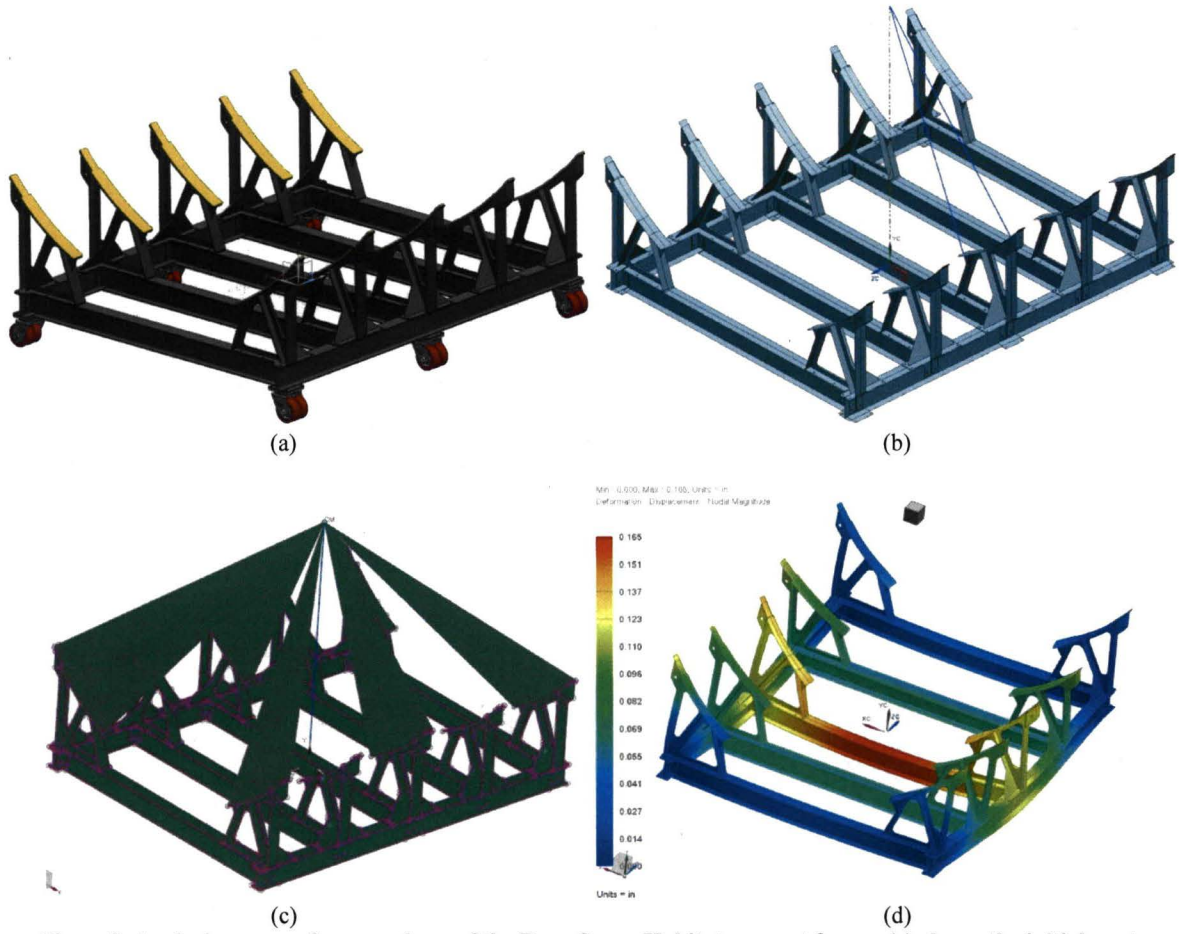
(a)

(b)

(c)

(d)

Figure 9. **Analysis process for a mock up of the Deep Space Habitat support frame. (a) shows the initial part, (b) shows the ipart, (c) shows the FEM, and (d) shows the displacement results.**