Modeling and Preliminary Finite Element Analysis on the Spun Structure for the CPInterSEP Project

A Senior Project

presented to

the Faculty of the Aerospace Engineering Department

California Polytechnic State University, San Luis Obispo

In Partial Fulfillment

of the Requirements for the Degree

Aerospace Engineering Bachelor of Science

by

Jason Carpenter
Kelly Cheng
Jeffrey Ma
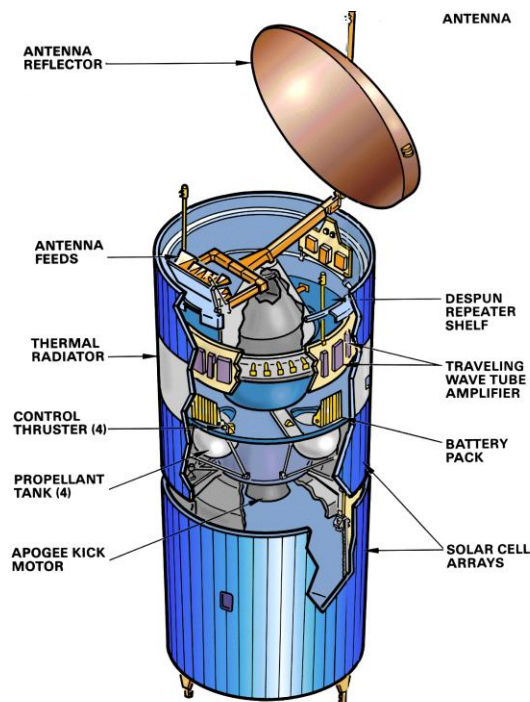Richard Pelham
Kevin Povey

December 22, 2010

Table of Contents

I. INTRODUCTION [Kevin Povey]

The Cal Poly Interdisciplinary Satellite Engineering Project (or CPInterSEP) is a prime example of Cal Poly's motto of "Learn by Doing". The project involves students, faculty, and industry personnel from various areas of expertise coming together to put a full-size functioning satellite into space. Starting with a basic satellite bus structure, students will work together with members of industry to plan, design, manufacture, and assemble each of the different spacecraft subsystems. In order to accomplish this task, it is essential to have a solid understanding of the capabilities and requirements of the base structure and this knowledge comes from stress analysis. In order to have an efficient process for stress testing and calculations, it is very helpful and essential in today's industry to be able to perform finite element analysis (FEA) on the primary structure of the spacecraft. Since no 3D CAD models were available for this structure, the first step is to create the model and then document the process for performing FEA on this model. In this case, the structure of the spacecraft is the BS376 satellite bus.

The project was able to come to life through the generous donation of two 376 type satellite buses by the Boeing Company, similar to the bus structure shown in the completed satellite diagram below.

ANTENNA

ANTENNA
REFLECTOR

ANTENNA
FEEDS

DESPUN
REPEATER
SHELF

THERMAL
RADIATOR

TRAVELING
WAVE TUBE
AMPLIFIER

CONTROL
THRUSTER (4)

BATTERY
PACK

PROPELLANT
TANK (4)

APOGEE KICK
MOTOR

SOLAR CELL
ARRAYS

**BOEING 376
SPACECRAFT CONFIGURATION**

The model 376 satellite bus was originally designed and manufactured by Hughes Space and Communication Co. until the company was bought by Boeing in 2000, where Boeing Satellite Systems continued to manufacture and improve the 376 design. The 376 spacecraft has always been used as a communications satellite but it can accommodate a wide variety of payloads and multiple configurations. Though the payload configuration and mission capabilities are flexible, all 376 spacecraft have a similar chassis design. All models have two telescoping cylindrical solar panels with body mounted solar cells where, once in orbit, the outer cylinder deploys downward in a 'Dixie-cup' fashion in order to increase the solar panel area, as shown in the figure below.
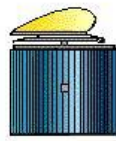
Satellite stowed and ready for launch
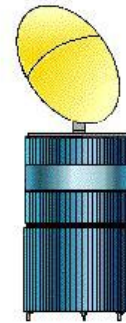
In orbit, fully deployed and ready to transmit

The spacecraft is spin-stabilized with a de-spun antenna platform and uses a hydrazine propulsion system for attitude control. The satellites using the 376 bus usually weighed around 1200kg (2600lb) with variation from program to program. The 376 bus design has proven to be one of the world's most-purchased commercial communications satellite models with almost 60 launches. The first was the SBS 1, which was launched on November 15, 1980 on a Delta 3910 from Cape Canaveral and the last was E-Bird, which was launched from Kourou on the Ariane 5G on September 27, 2003. Also of note, in 1983, a 376 spacecraft (Anik C3) was the first communication satellite to be launched from the space shuttle. These satellites can be boosted by any of the world's major launch vehicles and though there have been some failures, most are still orbiting today providing TV and radio coverage to 5 continents.

The most efficient and practical way to make a 3D CAD model of the 376 bus is through the use of detailed part and assembly drawings. The acquisition of these drawings, however, turned out to be much more difficult than expected. The first hurdle came from the fact that though the 376 was a popular model, it is no longer in production and all the documentation had been moved to storage. A Boeing employee would need to take time away from the work he or she needed to get done and go search through the warehouses for the drawings. This process itself took many weeks. The next hurdle came once the drawings were located. Since this information is ITAR controlled, it was not permissible to transfer the electronic data via the internet nor mail the physical copies through the postal system; the information needed to be handed over in person. It was an additional number of weeks before an arrangement was able to be made for a meeting. In addition, after the electronic versions of the drawings were acquired,

many were unable to be opened due to the need for Boeing in-house software. Finally, after the drawings had been obtained in an acceptable format, it was found that more than half of the drawings were still missing. As a result, the entire 376 satellite bus was unable to be modeled, instead, the available drawings were used to create a model of the majority of the spun portion only. In lieu of modeling the de-spun structure, much more effort was put into the task of performing the finite element analysis.

The primary purpose of this project is to provide a basis and groundwork for work to be done in the future. There will need to be a large amount of stress analysis done when comparing various payload packages or other possible additions to the spacecraft. In addition to creating a 3D CAD model of the spacecraft for visual purposes and as a base for future modeling, the project incorporates two other very important features. The project not only verifies that the FEA performed on the Pro/E solid model using Patran is accurate and gives expected results, but also provides a detailed, accurate, and user-friendly tutorial for performing FEA on Pro/E models using PATRAN and NASTRAN. Appendix A details the names and locations of the relevant files used in our project.

It is important to verify that results are similar to what is expected in order to ensure that the solid model is interacting appropriately with the FEA software. Very small incongruencies with the way a model is made or incorporated into the simulation can create drastic inaccuracies in the results. The FEA software will most usually give these incorrect results, with no message or indication that something is invalid. In order to verify that the model and simulation for the 376 spun structure is accurate, loads were analyzed individually in each of the 6 degrees of freedom, three in translation and three in rotation, and then compared to both the logical reactions and hand calculations. Due to the nature of the CPInterSEP, many other individuals in the future will need to perform similar stress testing and finite element analysis on a more complete satellite structure. In order to streamline this process and create a much shorter and less discouraging learning curve, the complete process from creating the models to viewing and analyzing the results needs to be documented in a format that can be followed by an inexperienced individual.

Overall, this project will be just the first steps in the legacy of CPInterSEP. It creates a foundation for the project to build upon and a starting point to go forward from. Each aspect of what has been accomplished should be able to help future participants both improve upon and expand the spacecraft. The project can be broken down into four main categories; creating the 3D model in Pro/Engineer based on the detail part drawings, modifying the existing model to be suitable for FEA, setting up the simulation model in Patran, and actually running the simulation model using Patran/Nastran.

References

http://www.selkirkshire.demon.co.uk/analoguesat/boeing376diagram.html

http://www.astronautix.com/craft/hs376.htm

http://www.boeing.com/defense-space/space/bss/factsheets/376/376fleet.html

## II. Best Practice Discussion [Kelly Cheng & Richard Pelham]

As our task will be to generate 3-D solid models for the primary structure, it would be beneficial to lay the groundwork for a set of rules for proper modeling practices. Proceeding in this manner would reduce incompatibilities of models between the different modelers and efficiently generate parts that would adhere to a unified structure for assembly purposes.

Part Modeling Practices

- Choose a Part Modeling Philosophy
  - Top/Down approach – Start with the smallest mass that will encompass the whole part and remove material to refine detail on the part
  - Bottom/Up approach – Start with a core shape and add features to refine detail onto the part
- Choose and operate under a standard dimensioning system. SI? English?
- Always plan out logical sequences in creating a part addressing parent/child relationships. This will allow for a set of related features to be suppressed/resumed without also suppressing/resuming other unrelated features.
- Carefully select and think about how you define the sketch plane on which you draw. The reference plane for which you choose the desired orientation of view will affect which direction a plane or surface faces after the sketch is complete. The chosen reference plane must be perpendicular to the sketching plane.
- Create features individually as this will mitigate problems when suppressing or deleting any number of various features.
  - Use the insert mode to modify and add features. This will reduce model regeneration time by not requiring regeneration of features that have no real involvement with the modification. This will help with the order of feature creation as well as maintaining proper parent/child relationships.
  - Do not try to sketch multiple closed sections in sketch mode as each section should be a unique feature.
  - Total part features should be minimized by compacting similar features using the patterns and groups functions.
  - Cosmetic features operations (such as *rounds* and *chamfers*) should be left to be performed at the end of the fundamental model tree. They should not be added from the sketch feature.
- Good Sketcher Use
  - Use sketched centerlines and construction circles for proper dimensioning scheme.

- o Remove any weak dimensions before completing a sketch by adding constraints, adding strong dimensions or converting weak dimensions into strong dimensions.
  - o Many simple features rule: keep the number of dimensions in a sketch to a maximum of eight as a rule of thumb. If more are necessary to properly dimension a feature, create separate datum features first and then reference these datum features in the sketcher.
- References
  - o Pick references in a 3-D view to avoid errors in choosing exactly what reference constrains a feature. The message window should be read after the selection is made to validate this reference choice has been correctly selected.
  - o Choose only as many references as are necessary for a robust model that can change as a design evolves.
  - o Good reference selection: datums or planar surfaces are good, edges are bad and may disappear when cosmetic features like rounding modify the entity, base features would make more stable choices than later features.
  - o Use *Replace* rather than deleting a sketched identity to redefine a sketch while maintaining the same entity id number and avoid rerouting all the children features from the old identity to the new one.
  - o Always use reference curves and surfaces to create complex shapes so that complex reference parameters will always be available and no parameters will be lost when a function fails or causes aborts.
  - o Relations should be created that associate features when direct references are not feasible.
  - o To change a feature reference, use the *Edit References* method and roll back the part temporarily to when a specific feature was created. This way, no mistakes will be made to accidentally choose a new reference that is newer than a feature created after it in the model tree.
- Documentation
  - o Descriptive names should be assigned for datum planes to provide detail for reference selection. Descriptive names for features should be assigned in the model tree whenever possible.
- *Revolving* for a feature in a part can be used as long as the sketch drawing is fully enclosed during editing of the definition. Not enclosing the drawing should be avoided as it blocks the user from using the *Hole*, *Shell* and *Rib* tools later in feature creation on some parts. *Blends* can also be used when these features are planned to be used to modify the feature. Revolve is a faster feature creation process when these features are not necessary.
- Layers for Parts Creation
  - o Create and use setup files to predefine name and states for all layers. This is useful for multiple parts to be used on the same assembly.

- o Mapkey can streamline this sequence of processes when performing the same command sequences repeatedly. This is especially useful for the set up and configuration of the same layers on multiple parts.
- o Layers can be used to efficiently control the display of datum planes when assembling a large number of different parts such as in the figure below.
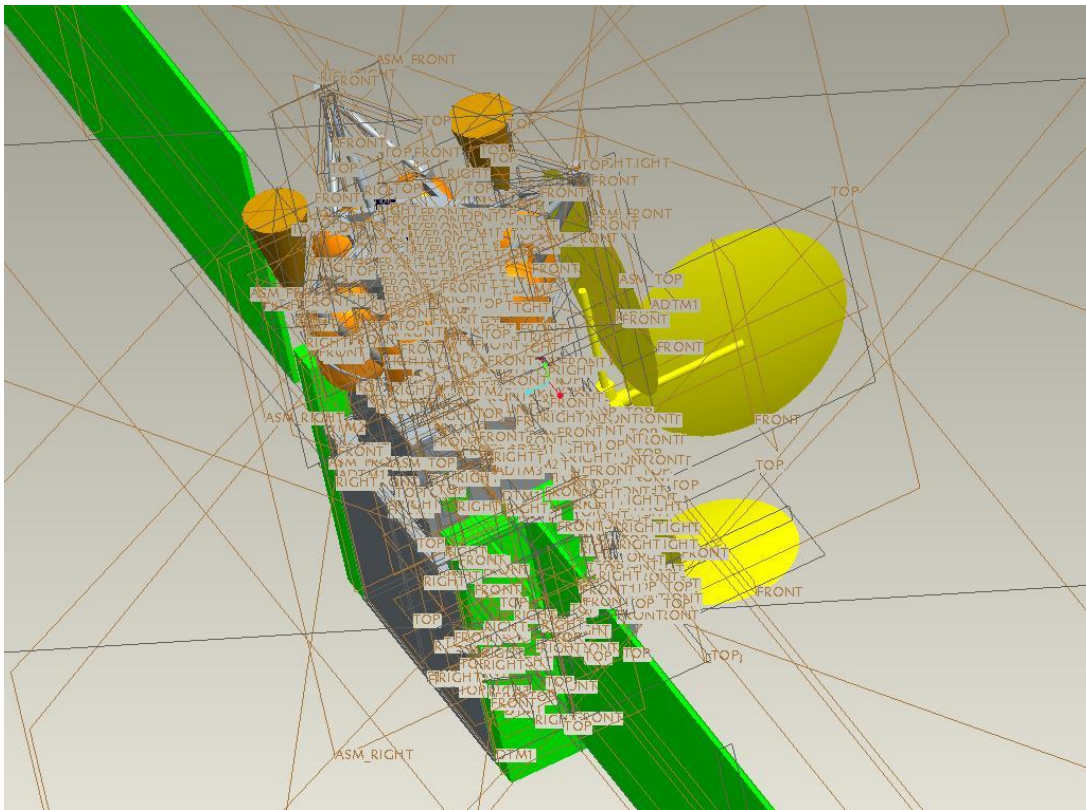
A video tutorial presentation to assist with layering in assembly models is presented at: http://www.ptc.com/appserver/wcms/replay/wmvPlayer.jsp?im_dbkey=68301&icg_dbkey=362

Assembly Practices

While building assemblies in Pro/Engineer, we will be using the following techniques to ensure each model is well built and constrained correctly.

- General Techniques
  - Use a top-down approach. Define where you are going from the start by looking at the project specifications. Define what you know at each level working from the top down.
  - Top assembly is cut into specific envelops. Sub assemblies do not limit top assemblies progress. Lead designer not responsible for maintaining volumes for sub assemblies, their locations and progress in top assembly.
  - Use skeletons, datums, and curves to define important areas and functions.
  - Use dummy parts as necessary to preserve important spaces.
  - Think about what you will need at each subassembly. Create those levels even if nothing is in them at the moment.
  - Do not get into details too early. This will save a lot of time and redefinition.
  - Bottom up design. Sub assemblies are added as they are completed with no area reserved for sub assemblies. Lead designer is responsible for maintaining top assembly and must insure we have latest data from each designer.
  - Either top-down or bottom-up design will work so long as everyone follows the same philosophy.
  - Avoid having the same part intersected by assembly features from different levels.
  - Use layers to group like entities so that the assembly is not too cluttered.
  - Reference base part to global axis. All other parts should be referenced to the part it is attached too.
- Constraints
  - Geometric relation between any two parts has six degrees of freedom: 3 translational and 3 rotational. All DOFs must be constrained.
  - Think about how to use these constraints to construct the assembly.
  - Most parts can be assembled using 3 constraints; some can be done in two. Make sure pro/e automatic assumptions are turned off to fully constrain a part.
  - The order in which components are brought into the frame is more important than the constraints.
  - Constraints lead to hierarchical structure of the assembly.
- Clearance and Interference
  - Determine if the parts in the assembly have clearance or interference.
  - Use hidden or no hidden lines to check for interference. Interference may or may not be desirable.
  - Global interference can check the entire assembly.
- Scales in Assemblies

o Small scale parts in large scale assemblies can be tedious to handle. Focus on the small scale part first, provide all the constraints for it, then go to the large scale part and place all the constraints for it. Pro/E allows you to fix constraints in different orders, and will assume a constraint for each incomplete constraint.

Integration between Pro/Engineer and Patran

- Geometry generated in Pro/E can be transferred directly to Patran. Save part or assembly as a step file in Pro/E. Import into Patran.
- When importing into Patran, Parasolid is an option to render the model. Parasolid models are "dumb" with no features or sketches. This is essentially what we want when dealing with FEA.
- By default Parasolid always uses metric in millimeters not meters. Must convert units upon import.
- Objects created in Pro/Engineer are General Trimmed Surfaces. Importing geometry with will require use of the tetmesh option in Patran in order to mesh correctly.

References

http://www.synthx.com/tom/sy_tips.htm - Library of Pro/Engineer Tips

http://www.mcaduser.com/ - Library of Pro/Engineer Tips

http://www.caddigest.com/subjects/pro_engineer/tutorials_proe.htm - Library of Pro/E Tips

http://www.imakenews.com/ptcexpress/e_article001430549.cfm?x=bfvRfnG,b3jsqcsB,w – Geometric Tolerance in Pro/E [see www.engineersedge.com/gdt.htm for a list of Geometric Tolerance call-outs]

http://www.ptc.com/appserver/wcms/standards/textmultitextimg.jsp?im_dbkey=13764&icg_dbkey=382&icg_dbkey=382 – Flexible components in Pro/E

http://www.blbeach.com/proe_tips.html

http://www.iheartrobotics.com/2009/02/proe-wildfire-tips-and-tricks.html

2008_CON_3D_Design_Practices_Samuels.ppt

http://www.sciencedirect.com/science?_ob=ArticleURL&_udi=B6V2D-49M6W42-2&_user=521828&_coverDate=01%2F31%2F2004&_rdoc=1&_fmt=high&_orig=search&_sort=d&_docanchor=&view=c&_searchStrId=1248708095&_rerunOrigin=google&_acct=C000059579&_version=1&_urlVersion=0&_userid=521828&md5=bc45b34ee6a65c53380f67f5dae2b69a#toc17 -

http://www.imakenews.com/ptcexpress/e_article000357760.cfm?x=b4nT7KG,b29hVpfK,w – Optimizing Memory Usage for Assemblies

http://www.imakenews.com/ptcexpress/e_article000404998.cfm?x=b11,0,w – Checking for Clearance in an Assembly

http://www.synthx.com/tom/sy_tip_0506.htm - Handling Scales in Assembly

http://www.imakenews.com/ptcexpress/e_article000899504.cfm?x=b11,0,w – Decreasing Assembly Picks

http://www.synthx.com/tom/sy_tip_0604.htm

http://www.imakenews.com/ptcexpress/e_article000357760.cfm?x=b4nT7KG,b29hVpfK,w

http://www.imakenews.com/ptcexpress/e_article000404998.cfm?x=b11,0,w

http://www.synthx.com/tom/sy_tip_0506.htm

PowerPoint 3D Design and Practices

Pro/Engineer Wildfire ™ 3.0 Tutorial

http://www-h.eng.cam.ac.uk/help/amb/programs/fe/prepost/patfaq/patranf1.html#Q1.5

http://people.msoe.edu/~rizza/FAQ_for_MSC.html

http://www-h.eng.cam.ac.uk/help/amb/programs/fe/prepost/patfaq/patranf6.html#Q6.13

III. PRO/ENGINEER MODEL [Jeffrey Ma]

Parts list

From Boeing we received an indentured parts list. The parts list included the part level, part number, organization name and a nomenclature. The part level is to indicate what parts go in a subassembly whereas the part number identifies the part for quick reference. The organization name is used to identify the part under the organization in which it was manufactured from. The nomenclature describes what the part, or subassembly, is. To better manipulate the parts list we created our own parts list. Our parts list included the part number, part level and nomenclature but we also added an assignment column and a notes section. The assignment listed who the specific part was assigned to and whether or not the assignment was completed. A completed assignment would be colored green whereas an assignment 'still-in-progress' would be colored orange. The notes section is to write down any notes about the specific drawing. Each group member was responsible for updating the parts list.

Our parts list proved to be very useful. It allowed us to know who worked on which drawing so that no two duplicated parts were generated by different team members. Also, the list allowed us to locate the part number easily as well as checking the description which allowed us to confirm the matching drawing. Another aspect of the parts list that helped was the part level. Instead of looking at many different assembly drawings, the part level allowed us to visualize the assembly and put together subassemblies.

Spun Structure

As the project went on, we started to receive drawings for the spun structure only. Once we received these drawings we distributed the drawings to each group member. The more difficult drawings were assigned to the more experienced modelers. However, everyone had their fair share of drawings and difficulties.

For this project we have built two different models. One model is for display purposes only, whereas the other model is only used for analysis. The display model has every detail drawn into it. Every ground strap, shim and thread will appear in the display model. For the analysis model there will only be structure that is load bearing.
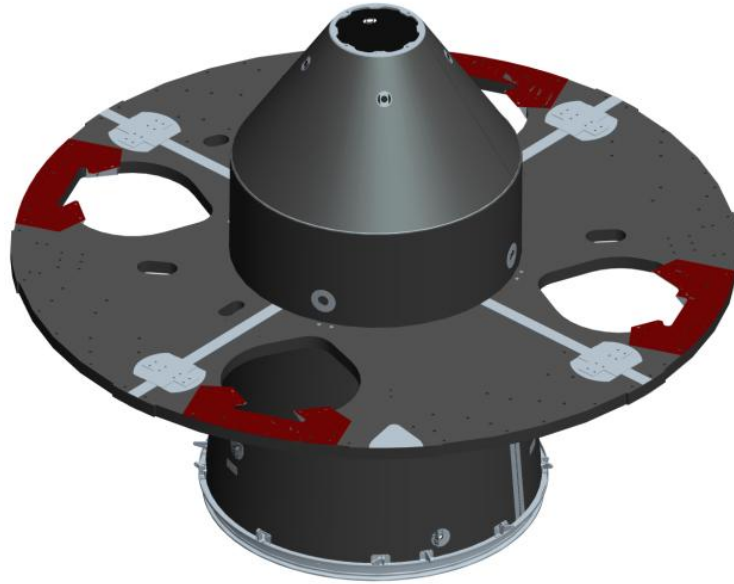
The analysis model will contain the four sections, 1) a cone, 2) a cylinder and two honeycomb sections. In addition to the four sections there are five rings that will not be imported from Pro/Engineer. These rings are the forward, intermediate, Apogee Kick Motor (AKM), Reaction Control System (RCS) and the separation ring.
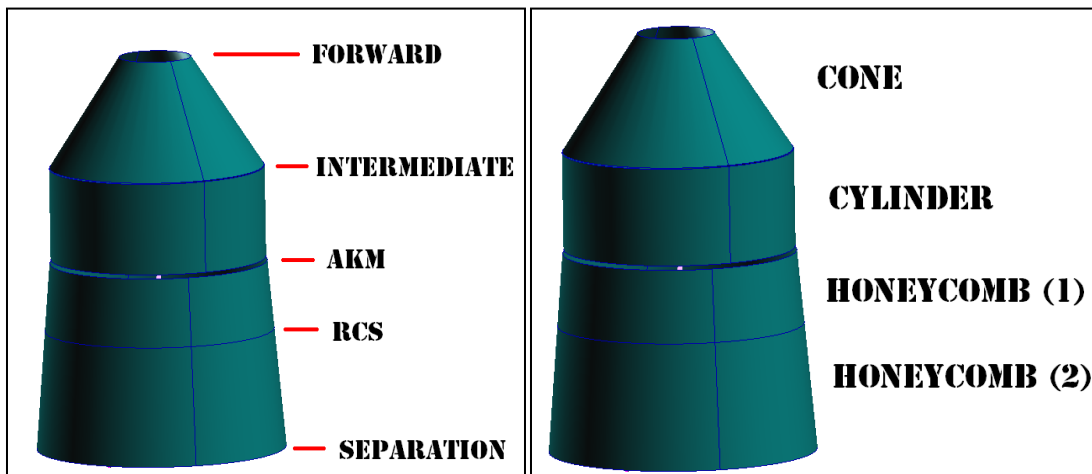
IV. PREPARING MODEL FOR FEA [Richard Pelham]

Rings

Rings are not actually being used in PATRAN. Instead, bar elements are used to simulate each ring. An element in PATRAN is generated and then rotated around the axis of preference. These elements must be in the correct location and have the correct material properties and moments of inertia associated with the actual structure. From the Pro/E model, the position and moment of inertia of each of the rings was determined by taking the initial sketch of each ring (the cross sectional area), extruding the sketch on both sides of the sketch plane, and using the x-section mass properties option. This option is located in the Analysis → Model tab. The radial center of gravity (CG) of the ring was used as the center of the bar element in PATRAN. The bar elements in the FEA model are located at the radial CG location for the physical ring it replaced. Material data sheets were used to find the material properties for each ring.

Skins

All of the skins used in PATRAN were modeled as surfaces at their neutral axis. The forward and intermediate skins are homogenous material, which means that the actual material thickness is also used for FEA. For the honeycomb skin, an equivalent thickness, density and Young's Modulus had to be calculated. The following equation was used to determine the equivalent core thickness,

$$\overline{}$$

where t_c is the actual thickness of the honeycomb, and t_1 and t_2 are the actual thicknesses of the face sheets. The following equation was used to determine the equivalent thickness,

$$- \quad \overline{}$$

where   is the equivalent core thickness, and t_1 and t_2 are the face sheet thicknesses. The equivalent thickness was found to be 0.4364 in. We found the equivalent material volume is 654.5956 in^3. This gave us an equivalent density of 0.006379 lb/in^3.
The equivalent Young's Modulus was found using the following equation,

$$\overline{}$$

where E is the actual Young's Modulus for the face sheets, and t_1 and t_2 are the face sheet thicknesses. This gave us an equivalent Young's Modulus of .60724E6.

V. PATRAN/FEA TUTORIAL [Kelly Cheng]

Background & General User Tips

In order to perform a finite-element analysis (FEA) on the thrust tube assembly of the spacecraft spun-section, the MSC FEA software suite was selected due to on-site availability and industry heritage. This section provides a step-by-step instruction guide for importing the equivalent thrust cylinder solid model, setting it up for test loads and accessing the results for plotting.

NOTE: Patran version 2008r2 was the latest version of MSC Software's pre/post-processing software available and was utilized for all work completed in this tutorial. Figures used in this tutorial represent a Patran interface that may vary for other versions of the software.

A lot of notes like the one above have been included throughout the tutorial. Though these are not necessary to build the FEA model, reading these will help you understand the concepts behind why you are setting up the model the way you are. Please read this for a better background on the model and the Patran/Nastran software suite. More often than not, an error down the line could have been avoided earlier on by understanding <u>why</u> you are doing something rather than just blindly following this tutorial and making a minor mistake.

The following is a brief summary of the workflow between Patran and Nastran, both of which will be utilized for our thrust tube analysis. Each step will be discussed in much more detail later in the report. The figure on the right details the overall workflow of the MSC software suite.[1]

- The basic geometry created in Pro/Engineer and described in the previous section will first be imported into Patran.
- Then, a finite-element mesh will be developed over the imported model.
- Several test cases consisting of various boundary conditions and test loads will be inputted.
- The proper material characteristics will be assigned to the proper shell and bar-element regions.
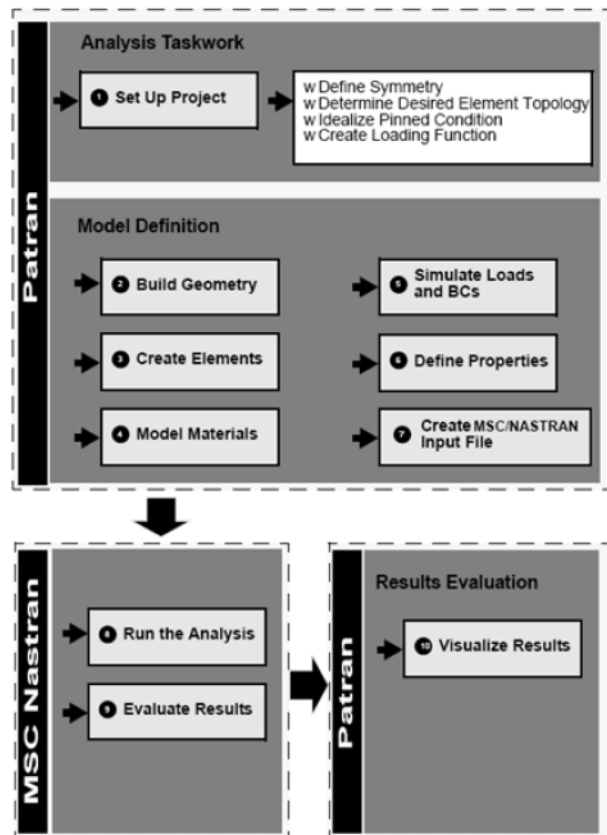
- The pre-processed model will then be sent into the MD Nastran solver for computation.
- The post-processed test results document can then be read by Patran and visual plots of the test report can be viewed for proper analysis.

The equivalent geometry to be used for finite-element analysis (FEA) purposes will primarily consist of the 2D shell surfaces representing the three skin sections and 1D bar elements representing the rings between each skin section.

The primary structure geometry created in Pro/Engineer will be imported into the MSC software suite in the IGES file format, **.igs**. The **.igs** neutral file format allows us to deliver geometries created in Pro/Engineer into Patran for use. This is necessary as we do not have the "ProEngineer Access" license for Patran to directly open model files.

$#   An "Patran 2008r2 ProENGINEER Access" license is not enabled. Please contact your MSC.Software representative for details on how to enable this functionality.

Property variations exist between **.igs** and other neutral formats such as **.step**. A whole chapter alone could be used to describe the nuances specific to each format. Simply put, **.igs** files operate correctly for our purposes on the type of equivalent geometry (shells, bars) that we have chosen for our model.

Initially, the newer **.step** file type was used though we did not require the extra product information architecture aside from the geometry information.[2] However, **.step** files began operating erratically when we switch from 3D solid geometry to 2D surface geometry as shown in the figure on the next page. When opened in PATRAN, these files would result in exploded shapes with strange curves. Thus, we returned to the older, more stable **.igs** files for the rest of this project.
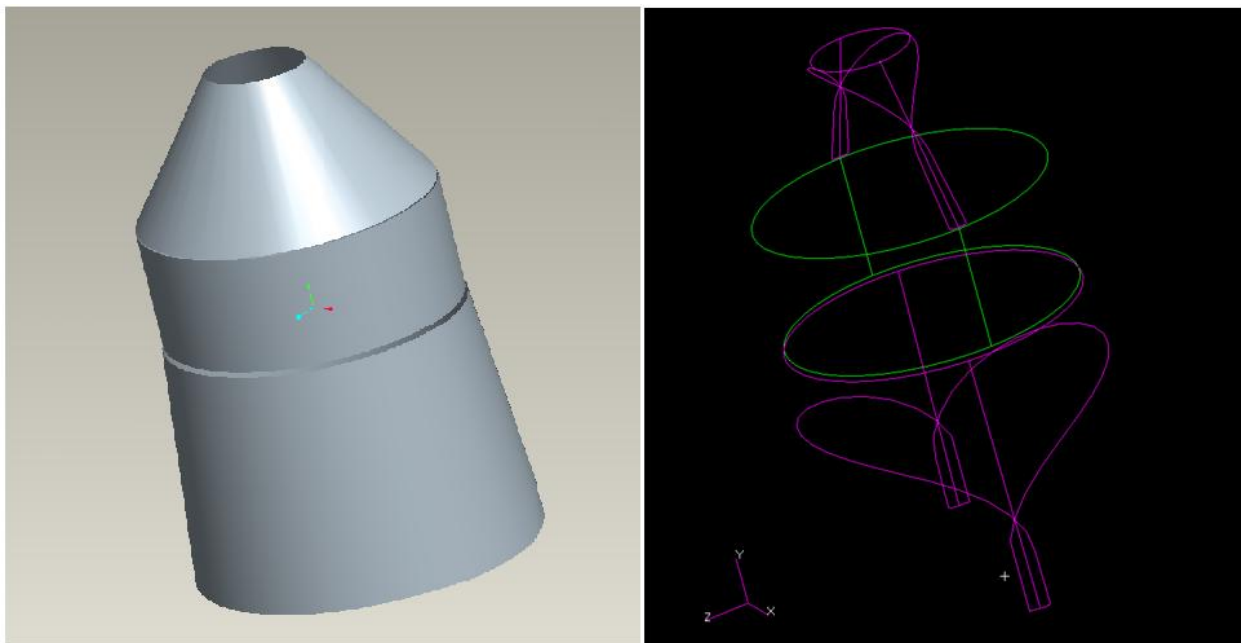
There are a few good practices to bear in mind when utilizing the Patran software in order to work more efficiently.

- Whenever an Application form is selected or any panel window is opened up, an **Auto Execute** box may be available for selection. Always ensure that this option is turned off. Otherwise, when any point, node, etc. is selected, the function may be applied without the user noticing causing confusion as well as unintentional changes to the model.
- After certain objects are deleted or un-selected, the primary model graphic window may not display the change until the graphics are refreshed. This can be accomplished simply by rotating and adjusting the view of the model.
- When possible, give unique IDs to your points and nodes in a method that will be easy for you to identify. Take note of this ID or the range of IDs that constitute a specific portion of the overall assembly model. This will provide for easier reference to appropriately select points, nodes or elements on the screen, when necessary.

Now we have laid down some general rules and guidelines to follow when using this software suite. We will begin going into detail regarding the process of setting up and analyzing the FEA model specific to our project.

FEA Modeling – Starting

Begin by accessing the MSC Patran 2008 r2 application and opening up a database file.

1. **Opening the Application**
   Click **Start → All Programs → MSC.Software → Patran 2008 r2 → Patran 2008 r2**
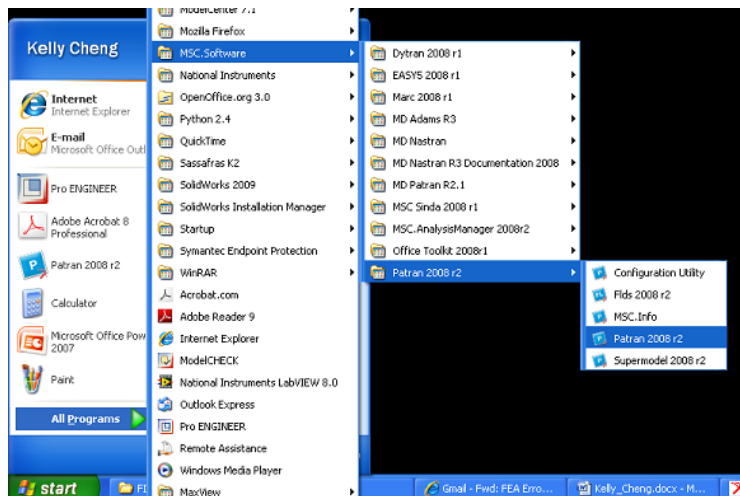
   Note: Exact Patran application location may vary depending on user setup and directory selection during program installation.

2. **Creating a New Database File**
   On the Main Menu bar, click **File → New**

   Select a directory location for the new database file and assign a file name. This database file will store all the physical model information, FEA mesh information, and any test Loads/BCs (boundary conditions) you have set.
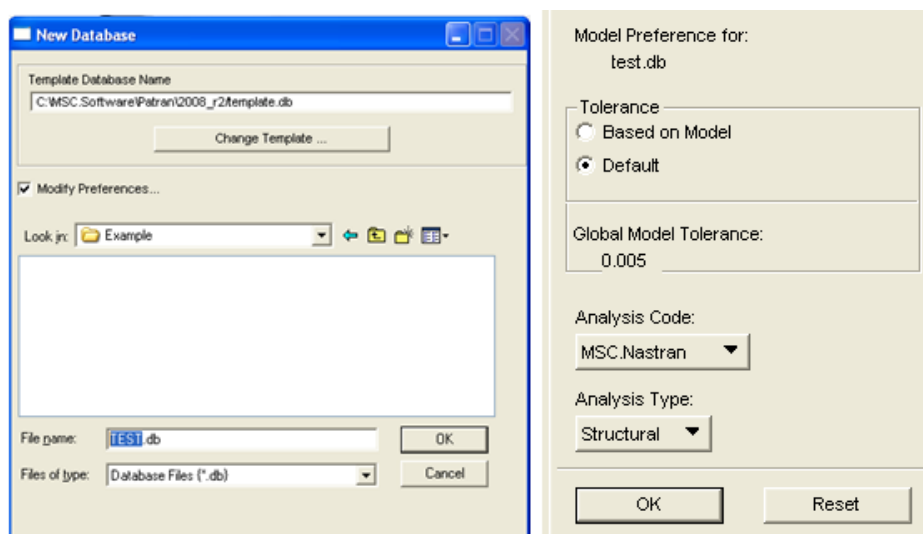
Set database **Tolerance → Default**
Set **Analysis Code → MSC.Nastran**
Set **Analysis Type → Structural**
Click **OK** to close the selection box.

3. **Importing the Geometry**
   Click **File → Import**
   Set **Object → Model**
   Set **Source → IGES**

   Note: As previously explained, we will be importing geometry from Pro/Engineer using the IGES neutral file format.

   Search your local directory until you are in your parts folder. Find the IGES file containing the thrust tube assembly model and select it.
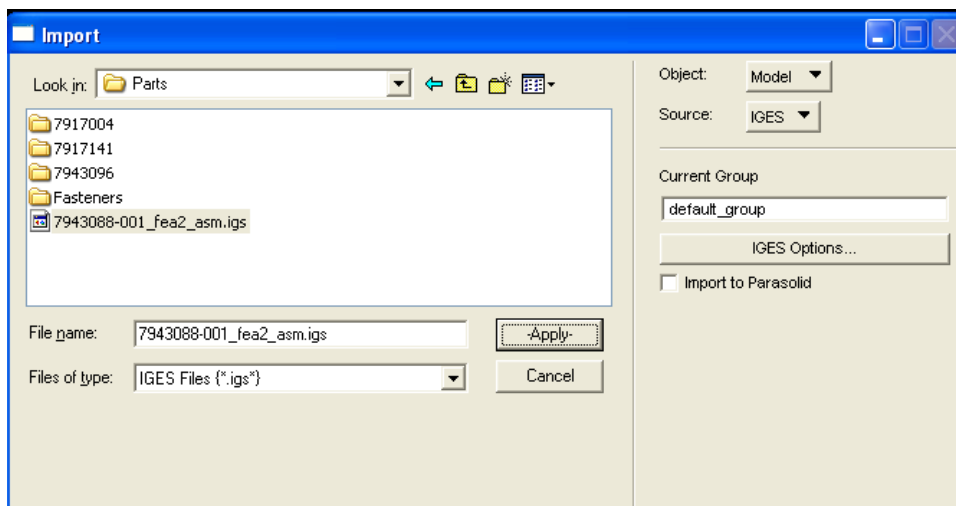
   Note: Please check to ensure that this is the equivalent 2D surface model version of the thrust tube assembly as detailed in the previous section.

   Click **Apply**.

   A successful IGES model import should result with the display of the assembly geometry in a green wireframe format.

FEA Modeling – Setting up the Analysis Model Parameters

4. **Creating a Cylindrical Coordinate System**
    Select the **Geometry** Application form from the Patran Main form.
    Set **Action → Create**
    Set **Object → Coord**
    Set **Method → 3Point**
    Set **Type → Cylindrical**
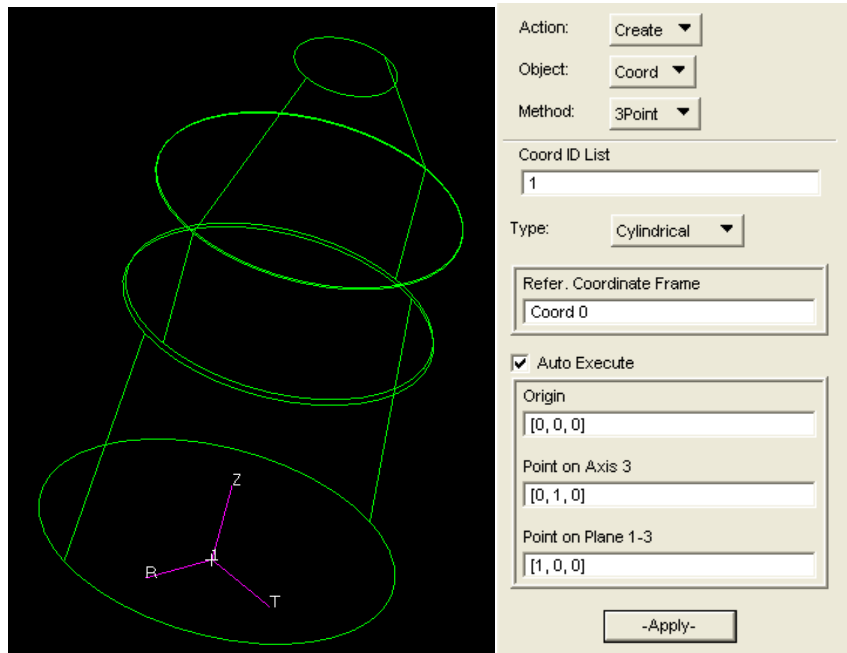
Note: The use of a cylindrical coordinate system will allow us to more easily define nodes and loading conditions relative to the imported spacecraft geometry later in the tutorial.

Set **Origin** to [0, 0, 0]
Set **Point on Axis 3** to [0, 1, 0]
Set **Point on Plane 1-3** to [1, 0, 0]
Click **Apply**.

This will create a new cylindrical coordinate system (r,Θ,z) under the label "Coord 1" with the origin located at the true location of the center of the bottom separation ring. This will be spatially lower than the equivalent bar element representing the separation ring to be created later. We will use this new coordinate system frequently as it will often provide an easier frame of reference as we build our model.

5. **Appending RCS Ring Cut to Imported Model Geometry**

The current imported model should have 6 surfaces representing three different sections on the spacecraft. In order to model the RCS ring axial location on the spacecraft, we will split a section's geometry into two to simplify locating and generating the ring elements. We will create a plane through the geometry to split the two sections.

This plane will be based on a generated point and vector to be explained here.

Select the **Geometry** Application form from the Patran Main form.
Set **Action → Create**
Set **Object → Point**
Set **Method → XYZ**

In **Reference Coordinate Frame**, type **Coord 1**. The reference coordinate frame for this point will be based about the cylindrical coordinate frame we created earlier.

In **Point Coordinates List**, set parameter to **[0 0 19.648].** This will generate a point on the axial (Z-axis) location of the structure where the RCS ring will be located.

Click **Apply** to finalize the point creation.

Set **Action → Create**
Set **Object → Vector**
Set **Method → Magnitude**

In **Reference Coordinate Frame**, type **Coord 1**. The coordinate frame will once again be with respect to the cylindrical coordinate system we created.

For **Vector Direction List**, set to **<0 0 1>**. This will set the vector to the axial Z-direction.

The **Vector Magnitude List** and **Base Point List** parameters should be left as their default settings, i.e. **1.0** and **[0 0 0]**.

Click **Apply** to finalize vector creation.

Now, we will use the point and the vectors to generate the plane with which to cut the lowest geometrical piece into two.
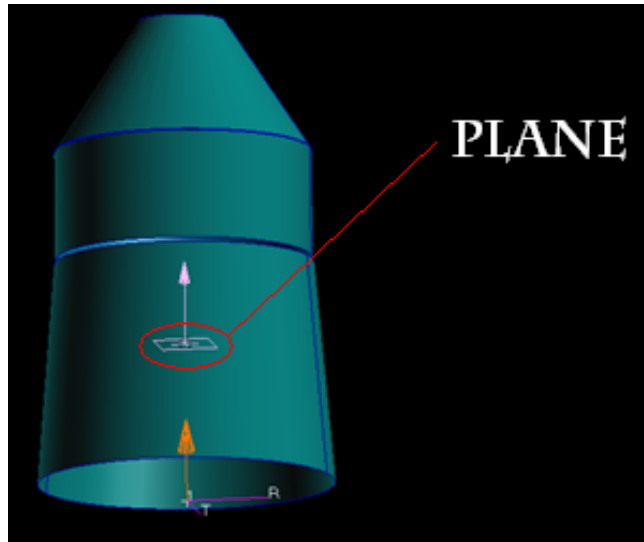
Set **Action → Create**
Set **Object → Plane**
Set **Method → Point-Vector**

In the **Point List** and **Vector List** selection boxes, select or type in the last two geometries we just created. These should be **Point 13** and **Vector 1** (if you have been following the tutorial in order).

Click **Apply** to generate the plane. The plane will be located normal to the Z-axial direction. Now we will use this plane to cut the bottom-third of the imported model.

Set **Action → Edit**
Set **Object → Surface**
Set **Method → Break**
Set **Option → Plane**

Ensure that **Delete Original Surfaces** is selected.

In **Surface List**, select the two surfaces located on the bottom third of the spacecraft or **Surface 1 2 (**if you have been following the tutorial in order). The two surfaces should be mirrored pieces that generate an expanding cylinder.

In **Break Plane List**, select the recently created plane or **Plane 1** (if you have been following the tutorial in order). This is the plane that Patran will use to break each of the old surfaces into two.

Click **Apply**.

A text prompt will appear asking if you are sure to delete the original surfaces. Select **Yes For All**, so that the newly created surfaces aren't overlapped by the old surfaces.

The successful alteration in the geometry should now result in eight total surfaces comprising our thrust tube assembly as shown in the image below.
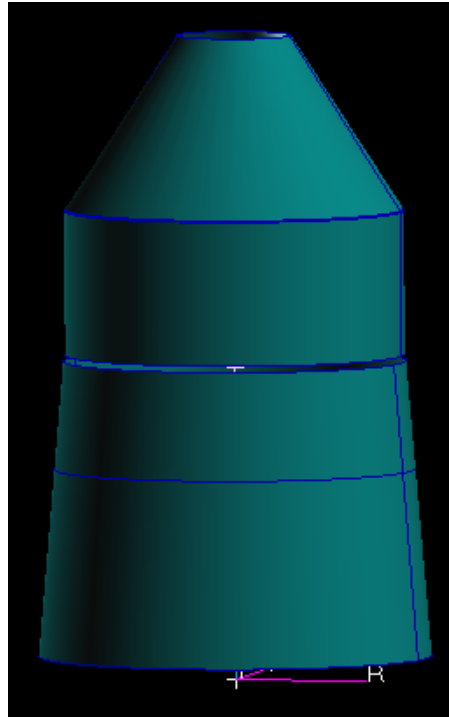
6. **Creating the Material Properties**

Select the **Materials** Application form from the Patran Main form.

For each isotropic material to be created, the same basic steps will be followed…
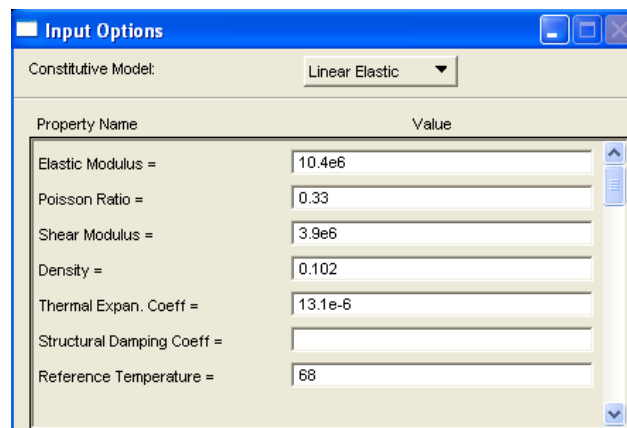
Set **Action → Create**
Set **Object → Isotropic**
Set **Method → Manual Input**

In the **Material Name** box, enter the name of the material.

Click **Input Properties …**

In the Input Options box, set the specific property values.

Enter relevant values for the **Elastic Modulus**, **Poisson Ratio**, **Shear Modulus** (if available), **Density**, **Thermal Expansion Coefficient** (if available) and **Reference Temperature**.
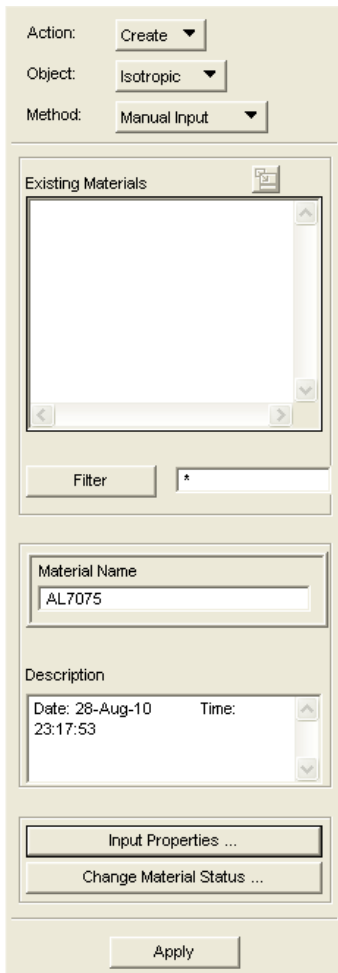
Click **OK** to close the **Input Options** box.

Click **Apply** to finalize the material creation.

Three isotropic materials and their properties will be set: **AL6061**, **AL7075**, and the equivalent **Honeycomb** material. The Aluminum Alloy properties can be found from various online sources and the equivalent **Honeycomb** material property was extracted from calculations made based on the equivalent material properties. They are listed in the table below (on the next page) for convenience.

	E, psi	γ	G, psi	ρ, lb/in^3	CTE,in/in-°F	T$_{ref}$,°F
AL6061	1.00E+07	0.33	3.77E+06	0.0975	1.31E-05	68
AL7075	1.04E+07	0.33	3.90E+06	0.1020	1.31E-05	68
Honeycomb	1.05E+07	0.33	4.00E+06	0.1010	1.31E-05	68

Table I – Spacecraft Relevant Material Properties

*Note: When creating the **Honeycomb** material, an error message will appear. Press **Yes** to continue. This message appears due to the fact that the material is not actually **Isotropic**. However, later we will modify the material properties to account for this fact.*
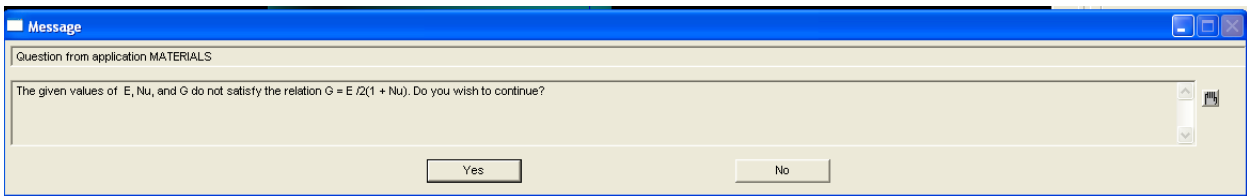
Repeat this process until all three **Isotropic** materials are created.

Next, we will generate the **2D Anisotropic** properties for the **Honeycomb** material.

Set **Action → Create**
Set **Object → 2D Anisotropic**
Set **Method → Manual Input**
In the **Material Name** box, type in the label **Honeycomb_anisotropic**.

Click **Input Properties** to bring up the various parameters to set for the material.

Enter the appropriate values for the material property, **G$_{ij}$**, matrix for the **Honeycomb** material.

In **Stiffness 11**, enter **27000**.
In **Stiffness 22**, enter **13000**.

Click **Ok**.
Click **Apply** to finalize this material creation.

Note: These values were obtained from calculations by Ray Ng, Boeing Space Systems. (raymond.w.ng@boeing.com, 310-364-5561)

FEA Modeling – Creating and Setting the Element Properties

7. **Creating the Assembly Rings BAR Elements, Method (A)**
   We will need to model five different ring structures that are fixed onto the skin currently modeled. These rings are on different Z-axial locations with respect to our cylindrical coordinate system.

   The rings will be designated as follows (from the forward-direction to the aft): **Forward**, **Intermediate**, **AKM**, **RCS** and **Separation**. The figure on the below details these designations.
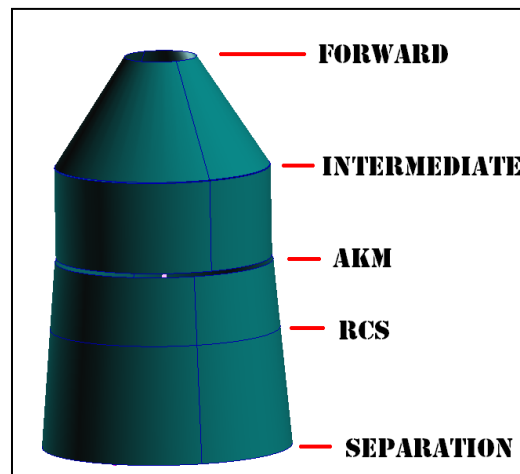
   All five rings will be modeled using **Bar**-elements. However, due to the nature of the geometry, two different methods will be utilized to fully model all five rings. Though they will behave similarly in the final model, the method in which the **Bar**-element rings will attach to the skins will vary.

   First, a **Bar**-element ring will be overlaid right onto the geometry for three of the rings (**Forward**, **RCS**, and **Separation**) and, where appropriate, will have offset node vectors to simulate the behavior of the rings from their true location. We will designate this **Method A**.

   As the other two rings (**Intermediate** and **AKM**) exist at Z-axial locations between skin sections, they will have **Bar**-element rings created in floating space, not touching any geometry. These rings will not need any offset node vectors as they already exist on the neutral axis of the true ring location. We will designate this **Method B**.

We will generate the three overlaid rings first using **Method A**. The rings to be created are highlighted in the figure below.
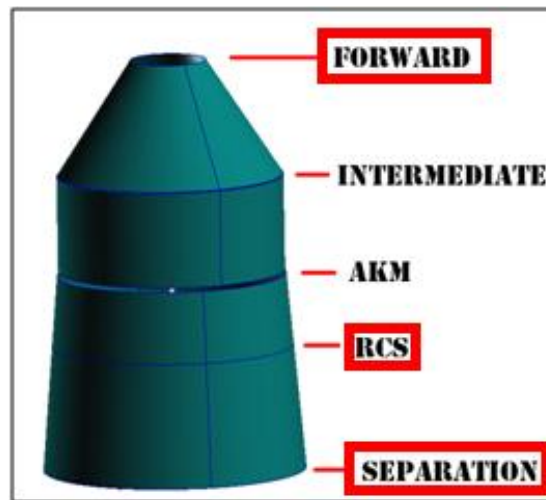
Begin by creating the nodes from the top layer of the **Cone**.

Select the **Elements** Application Form from the Patran Main form.

Set **Action → Create**
Set **Object → Node**
Set **Method → Edit**

Ensure that **Associate with Geometry** has been selected.

In **Analysis Coordinate Frame** and **Coordinate Frame**, select or enter **Coord 1** to set the nodes to the cylindrical coordinate system.

In the **Node Location List**, select or enter one of the point locations on the Z-axial skin location appropriate to the ring you are creating. This will be **Point 11/Point 12** on the top level of the cone for the **Forward** ring (if you have been following the tutorial in order).

Click **Apply** to finalize the node creation.

To generate our first element, we will need at least two nodes. We have just created the first node. The second node will simply be a rotational transformation of the first node.

Set **Action → Transform**

Set **Object → Node**
Set **Method → Rotate**

In **Refer. Coordinate Frame**, type **Coord 1**.

In **Axis**, type **Coord 1.2**. This will allow for proper Z-axis node rotation based on our current coordinate system settings.

*Note: By connecting two nodes, we can generate a **Bar**-type element that will act as the rigid connection tying several node couples together. After generating a total of 72 connected (each spaced five-degrees apart) bar elements, they will act as a rigid structure and simulate a structural ring.*

In the **Rotation Parameters**, select **Rotation Angle** and enter **5**. Ensure **Offset Angle** is set to **0** and **Repeat Count** is set to **1**.

In the **Node List**, select or type in the first node you generated. This should be **Node 1** (if you have been following the tutorial in order).

Click **Apply** to generate the second new node. This will result in two nodes separated five-degrees apart from the center of rotation of the spacecraft.

The **Bar**-elements for our first ring, the **Forward** ring, will now be created using the two nodes we have just generated.

Set **Action → Create**
Set **Object → Element**
Set **Method → Edit**
Set **Shape → Bar**
Set **Topology → Bar2**
Set **Pattern → Standard**

We will need to generate the properties for the bar elements as they pertain to the characteristic of each ring.

Select **Create New Property**. Ensure that **Object** has been set to **1D** and **Type** has been set to **Beam**. Select the **Property Set Name** box and enter a proper title.

Select **Input Properties**.

In order to select the proper **Material Name**, click the [icon] icon and select the proper material type from the list. Refer to Table II below for the material type of each ring.

Method A	Material	Area, in^2	Inertia 1,1 (Iyy)	Inertia 2,2 (Ixx)	Inertia 1,2 (Ixy)
Forward Ring	AL7075	1.9554403E-01	2.0691743E-02	1.8052725E-02	1.7350595E-02
RCS Ring	AL6061	1.0116500E-01	1.2025430E-02	5.6087391E-03	8.2344979E-04
Separation Ring	AL7075	7.6807750E-01	6.0103882E-01	8.1954054E-02	-3.3276043E-02

Table II – Spacecraft Ring Method (A) Properties

Scroll down to the **Area** input box as well as the Inertia matrix inputs (**Inertia 1,1 2,2 and 2,1**). Enter the appropriate values based on the table above.

Select the **Bar Orientation** box and enter **<0,0,1>** in order to create the **Bar** elements with the proper orientation,.

Set both **Offset @ Node 1** and **Offset @ Node 2** to the vector value appropriate for the ring as shown in the table below. For example, the values **<-0.074807,0,0.169395>** would be inputted in this field for the **Forward** ring. All coordinate values are in the standard **Coord 0** rectangular coordinate frame.

Method A	X, in	Y, in	Z, in	Referenced from…
Forward Ring	-0.074807	0	0.169395	CONE (Top)
RCS Ring	-0.481865	0	0.000000	HONEYCOMB (1) (Bottom), (2) (Top)
Separation Ring	0.037298	0	-1.007419	HONEYCOMB (2) (Bottom)

Table III – Spacecraft Ring Method (A) Offset Locations[3]

The standard skin callouts are shown in the figure below.
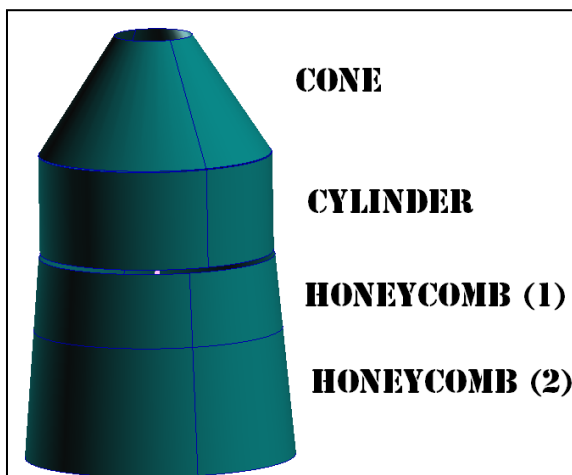
Click **Ok**. Select **Apply** to finalize your property creation.

For **Node 1**, select or type in the ID of one of the newly created nodes.
For **Node 2**, select or type in the ID of the other newly created node.

Click **Apply** to create the first **Bar** element, shown as a yellow curve in the displayed figure below.
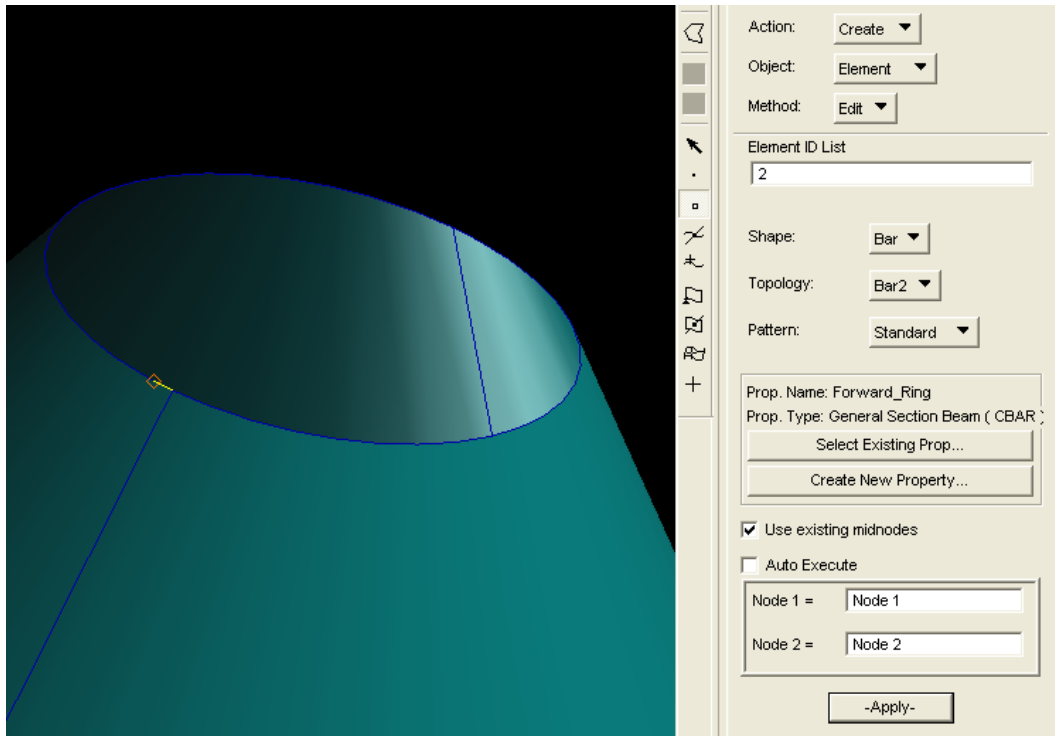
Now we will need to replicate this five-degree bar element 72-times in order to get the full circular ring.

Set **Action → Transform**
Set **Object → Element**
Set **Method → Rotate**

Once again set the **Refer. Coordinate Frame** to **Coord 1** and set the **Axis** to **Coord 1.2** for proper functionality.

In the **Rotation Parameters** box, set **Rotation Angle** to **5** for a five degree rotation again. Next, set **Repeat Count** to **71** as we will need 71 additional five-degree elements to generate a complete ring.

Click the **Element List** box and either select or type in the ID of the newly created **Bar** element.

Click **Apply** to generate the first full ring made up of 72 **Bar**-elements. An example of a successfully generated ring is shown in the figure below.
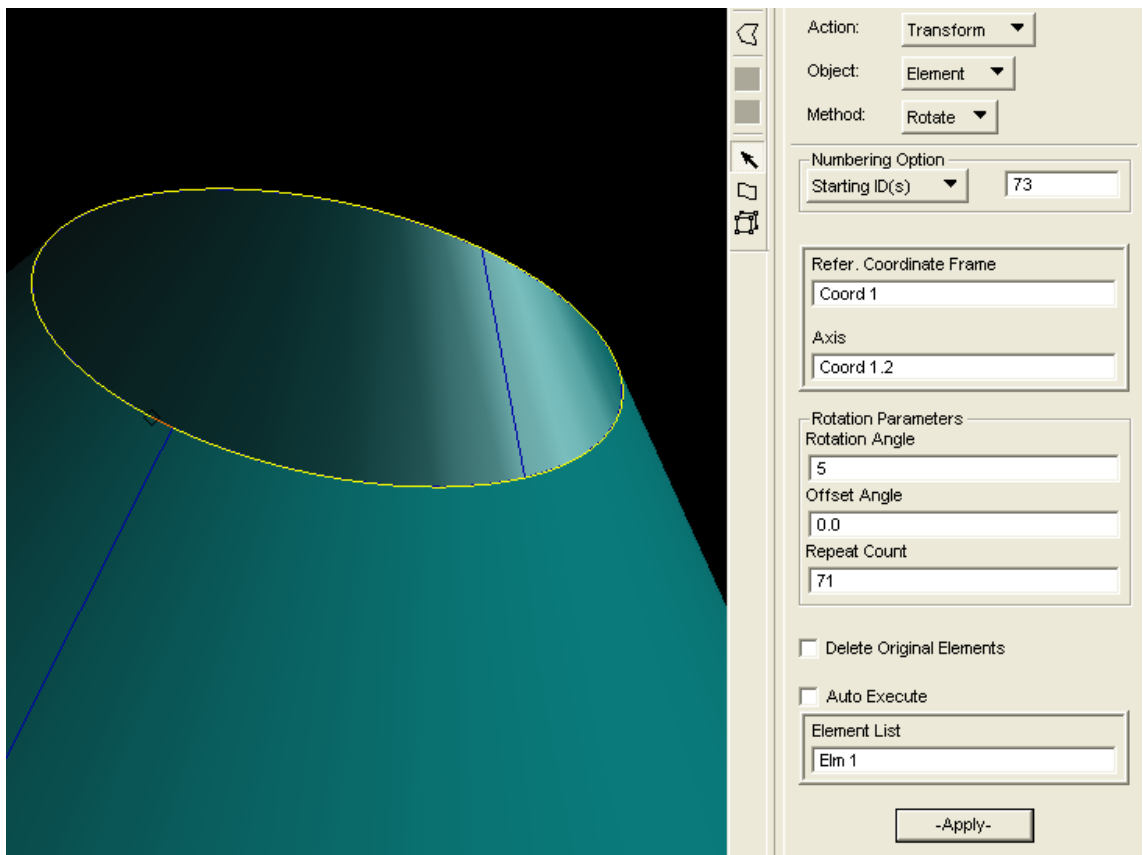
Note: You may notice that this will generate several overlapping nodes. However, we can take care of all the overlapping nodes and elements later using the equivalence function. Thus, we can ignore this issue for now.

Repeat the process for the **RCS** ring between **Honeycomb (1)** and **Honeycomb (2)** and again for the **Separation** ring at the bottom layer of the **Honeycomb (2)** section.

Although some of the configuration for generating and transforming the points and nodes will remain the same, slow down to check/ensure that all coordinate frames and other parameters are what they should be.

After all three rings structures have been placed on the assembly model, the model look similar to the figure below.
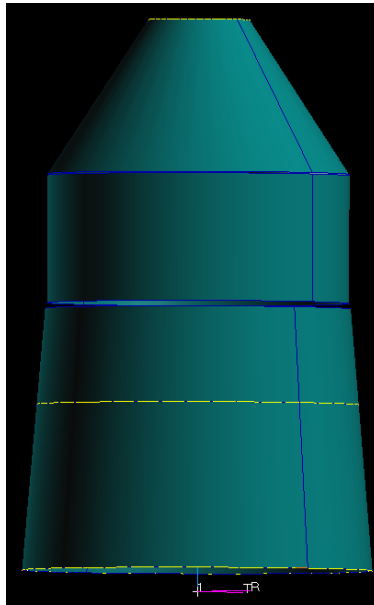
8. **Creating the Assembly Rings BAR Elements, Method (B)**

We will now generate the two remaining ring elements. The two remaining rings lie floating between the three main skin sections of the spacecraft.
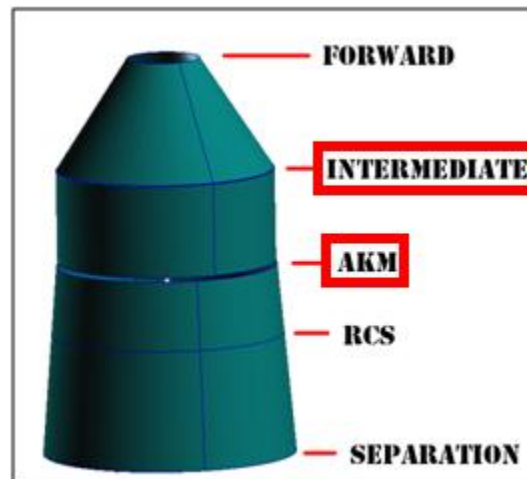
Select the **Geometry** Application Form from the Patran Main form.

First, it is necessary to create floating geometric points that will locate the nodes of the ring structure. We will do this by transforming points a specific distance from the edge of the ring's closest skin structure.

Set **Action → Transform**
Set **Object → Point**
Set **Method → Translate**

In **Type of Transformation**, set option to **Curvilinear in Reference Coordinate Frame**.

In **Reference Coordinate Frame**, type **Coord 1**. The reference coordinate frame for this point will be based about the cylindrical coordinate frame we created earlier.

In **Translation Vector**, an offset vector should be inserted based on which ring element the user is creating. The values for both remaining rings can be found in the following table.

Method B	X, in	Y, in	Z, in	Referenced from…
Intermediate Ring	-0.163711	0	0.034856	CYLINDER (Top)
AKM Ring	0.014453	0	-0.241312	CYLINDER (Bottom)

Table IV – Spacecraft Ring Method (B) Offset Locations [Reference 4.3]

Note: The imported solid model will have 4 points for the each of the three different skin layers (this is not exactly true with the honeycomb skin as we have further split that surface into multiple pieces). We will split and designate two of these points for the top level and two for the bottom level of each skin. You can see this in the figure below as each layer of the assembly skin is split into two halves of a cylinder. The corners of these geometries are what constitute the two points at the top and bottom of the skin section. As long as the correct point for the level is selected, it does not matter which of the two points you designate. The honeycomb section is split using one curve and so the two honeycomb layers should meet at the same points.

*Note: The Patran interfaces often acts strangely and does not allow you to use your mouse to select a point even though it is very much there. You will need to find the Point ID number using **Action → Show** and manually input it into the **Point List**.*

In **Point List**, we will select the point that the offset point will be generated from as shown in the "**Referenced from…**" column of the table above. The standard skin callouts used are once again displayed in the figure below.
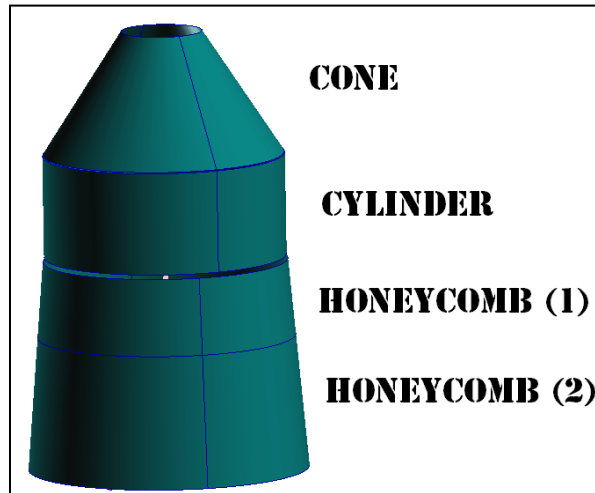
Click **Apply** to create the floating point which represents a point on the neutral axis of the yet-to-be created ring.
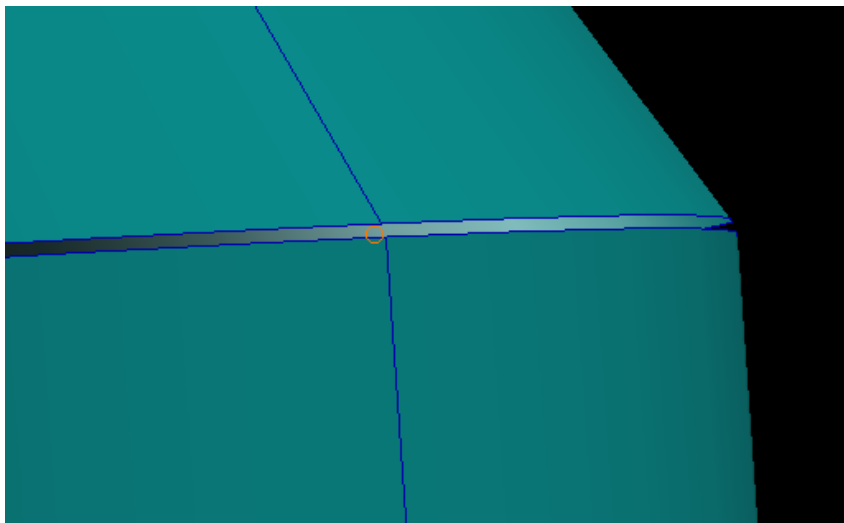
*Note: Keep track of the Point ID you have just created. Using **Action: Show**, **Object: Point** and **Info: Location,** you can verify that the point you have created is where it should be. Note the coordinate location of the point as well as the graphical location on the model.*

Now, we will create the node and bar elements for the ring from the newly created geometric point.

Select the **Elements** Application Form from the Patran Main form.

Set **Action → Create**
Set **Object → Node**
Set **Method → Edit**

For both **Analysis Coordinate Frame** and **Coordinate Frame**, type **Coord 1**. Now all analysis and coordinate frames will for these rings will automatically be set in the cylindrical coordinate system.

In **Node Location List**, select or type in your newly created point. This should be **Point 16** (if you have been following the tutorial in order).

Click **Apply** to generate the new node.

To generate our first element, we will once again need to make a rotational transformation of the first node. This will generate the second node and give us two nodes to make the element with.

Set **Action → Transform**
Set **Object → Node**
Set **Method → Rotate**

In **Refer. Coordinate Frame**, type **Coord 1**.

In **Axis**, type **Coord 1.2**. This will allow for proper Z-axis node rotation based on our current coordinate system settings.

In the **Rotation Parameters**, select **Rotation Angle** and enter **5**. Ensure **Offset Angle** is set to **0** and **Repeat Count** is set to **1**.

In the **Node List**, select or type in the first node you generated.

Click **Apply**.

35

This will create two nodes separated five-degrees apart from the center of rotation for the spacecraft.

*Note: By connecting these two nodes, we can generate a **Bar**-type element that will act as the rigid connection tying several node couples together. After generating a total of 72 connected (each spaced five-degrees apart) bar elements, they will act as a rigid structure and simulate a structural ring.*

Set **Action → Create**
Set **Object → Element**
Set **Method → Edit**
Set **Shape → Bar**
Set **Topology → Bar2**
Set **Pattern → Standard**

We will need to generate the properties for the bar elements as they pertain to the characteristic of each ring.

Select **Create New Property**. Ensure that **Object** has been set to **1D** and **Type** has been set to **Beam**. Select the **Property Set Name** box and enter a proper title.

Select **Input Properties**.

In order to select the proper **Material Name**, click the [icon] icon and select the proper material type from the list. Refer to Table III below for the material type of each ring.

Method B	Material	Area, in^2	Inertia 1,1 (Iyy)	Inertia 2,2 (Ixx)	Inertia 1,2 (Ixy)
Intermediate Ring	AL6061	9.7129774E-02	3.0130727E-03	2.5698090E-02	-7.7238827E-03
AKM Ring	AL7075	4.1226461E-01	5.6739987E-02	7.7814579E-02	9.5608966E-03

Table V – Spacecraft Ring Method (B) Properties

Scroll down to the **Area** input box as well as the Inertia matrix inputs (**Inertia 1,1 2,2 and 2,1**). Enter the appropriate values based on the table above.

Select the **Bar Orientation** box and enter **<0,0,1>** in order to create the **Bar** elements with the proper orientation,.

Set **Offset @ Node 1** and **Offset @ Node 2** both to **<0,0,0>**.

Note: Structural analysis pertaining to these two specific rings will be analyzed at the neutral axis where they have been already placed from the offset locations given. Thus, no offset values from the existing nodes need to be inputted in this field.

Click **Ok**. Select **Apply** to finalize your property creation.

For **Node 1**, select or type in the ID of one of the newly created nodes.
For **Node 2**, select or type in the ID of the other newly created node.

Click **Apply** to create the first **Bar** element shown as a yellow curve in the displayed figure below.
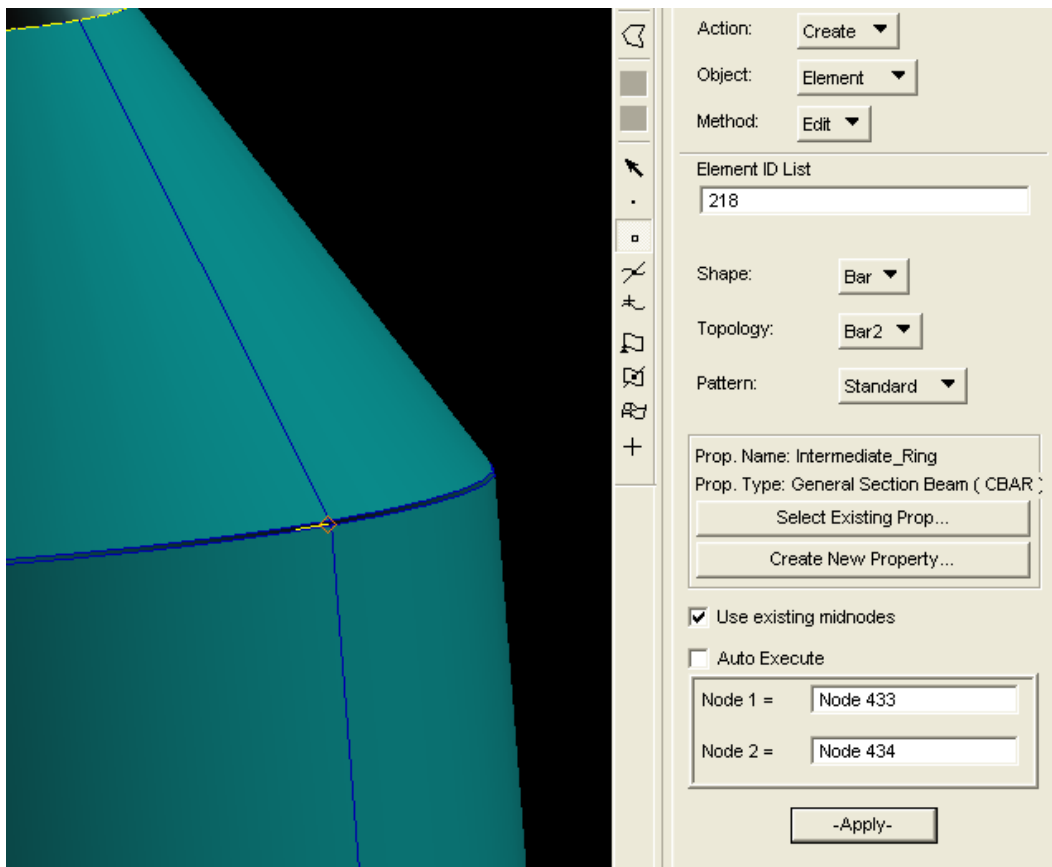
Similar, to the method used for the previous three rings, we will replicate this five-degree bar element 72-times in order to get the full circular ring.

Set **Action → Transform**
Set **Object → Element**
Set **Method → Rotate**

Once again set the **Refer. Coordinate Frame** to **Coord 1** and set the **Axis** to **Coord 1.2** for proper functionality.

In the **Rotation Parameters** box, set **Rotation Angle** to **5** for a five degree rotation again. Next, set **Repeat Count** to **71** as we will need 71 additional five-degree elements to generate a complete ring.

Click the **Element List** box and either select or type in the ID of the newly created **Bar** element.

Click **Apply** to generate the first full ring made up of 72 **Bar** elements. An example of a successfully generated floating ring is shown in the figure below.
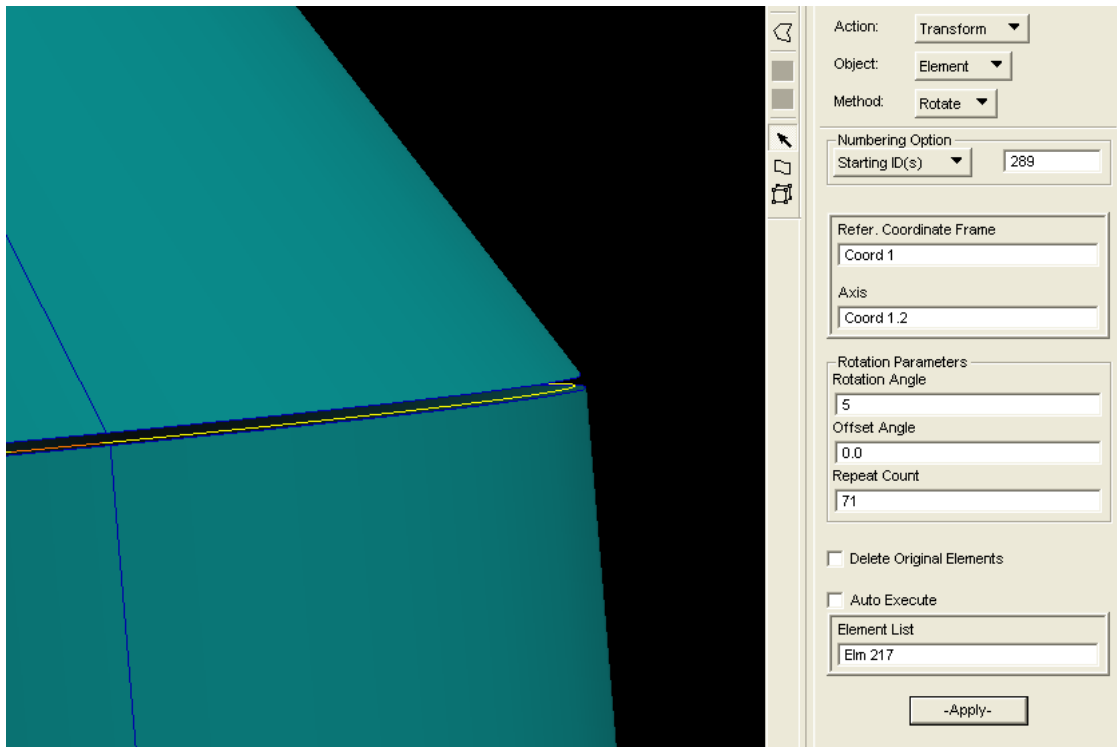
Note: You may notice that this will generate several overlapping nodes. However, we can take care of all the overlapping nodes and elements later using the equivalence function. Thus, we can ignore this issue for now.

Now, you will need to repeat this complete process for the other ring (the **AKM** ring) using its respective values. Although some of the configuration for generating and transforming the points and nodes will remain the same, check to ensure that all coordinate frames and other parameters are what they should be.

After all five rings have been successfully generated, the Patran assembly model will look like similar to the figure on the right.

Although all five sets of rings have been generated physically, using the **Transform** function to generate these elements does not also **Transform** the properties of the original element at each ring. However, now that we have generated each of the **Properties** for the five rings, we can re-designate the applicable region from one **Bar**-element to all 72-elements for each ring structure.
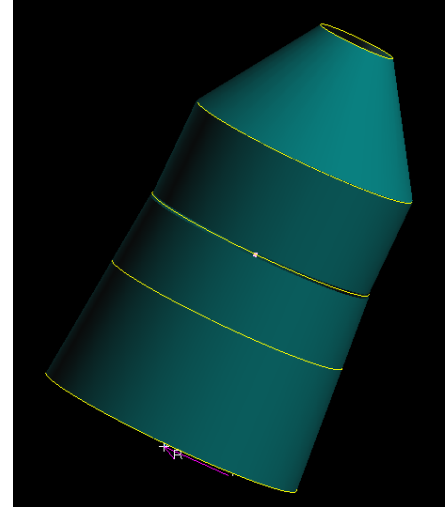
Select the **Properties** Application Form from the Patran Main form.

Set **Action → Modify**
Set **Object → 1D**
Set **Type → Beam**

In the **Sets By** list, select a specific ring which will open up the **Modify Properties** screen. Although we need to select the specific **Property Set Name**, we do not need to alter any of the previously defined properties. Select **Ok** to close this screen.

Click the **Select Application Region** box, to open up the applicable region for the selected property.

For each ring entity, there should only be one existing applicable region already inputted. We will need to <u>type</u> in and append the **Application Region** with the other 71-element IDs for each ring. For example, the **Forward Ring** will be modified from **Element 1** to **Element 1:72** (if you have been following the tutorial in order).

*Note: In the current version of the software, only **Surfaces**, not **Elements**, can be selected. Regardless, the **Element IDs** can be typed in manually.*

Click **Add** to append the newly selected members onto the original single **Element** ID.
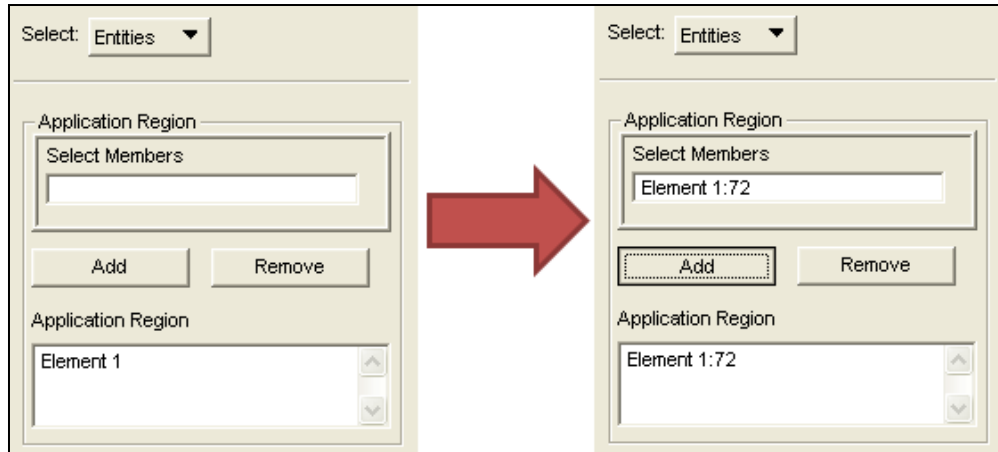
Click **Ok** to confirm the new selection.

Click **Apply** to confirm modification of all 72-elements with the selected property set.

Repeat this process for all five sets of rings. The table below will assist in entering **Element** IDs for each unique ring structure.

*Note: Depending on the order in which you created the rings, the Element IDs may be different in your case. These should be the appropriate ID values if you have been following this tutorial in order. Regardless each **Ring** should have exactly 72 **Element** IDs corresponding to it.*

	Element IDs
Forward Ring	1:72
Intermediate Ring	217:288
AKM Ring	289:360
RCS Ring	73:144
Separation Ring	145:216

Table VI – Ring-Specific Element IDs

*Note: After modifying the properties of these rings, take the time to use the **Action: Show**, **Object: Element**, **Info: Attributes** and select one of the non-original **Bar**-elements to ensure that the ring properties for all five rings have indeed been updated.*

9. **Creating the Skin Surface Mesh Elements**

Now to create the **Quad** mesh elements using a mesh seed to line the geometry. This mesh seed basically outlines how many interior mesh elements will exist for every curve.

*Note: **Quad** elements were chosen rather than triangular due to the rectangular overall shape of our structure. As such, **Quad** elements would provide for cleaner mesh seeding and a simple uniform mesh-element model.*

Select the **Elements** Application Form from the Patran Main form.
Set **Action → Create**
Set **Object → Mesh Seed**
Set **Type → Uniform**

*Note: The total number of mesh elements around the edge of every skin layer will need to match the total number of bar elements. 72 mesh seeds will allow for a five-degree rotational separation similar to the ring nodes we created earlier. As a full revolution of each edge of the four spacecraft skins (**Cone**, **Cylinder**, **Honeycomb(1)** and (**2**)) consists of two surfaces, we will create a mesh seed of length 36 for each of these curve-halves.*

Ensure that the **Number of Elements** button is selected.

In the **Number** entry box, type in **36**.

In the **Curve List**, select or type in the highest Z-axial curve-edge of the **Cone** skin. This should be either **Surface 5.3** or **Surface 6.3**.

Click **Apply** to generate the 36-element mesh seed.

These seeds should lie right on top of the individual nodes we transformed earlier to generate the **Bar**-elements.
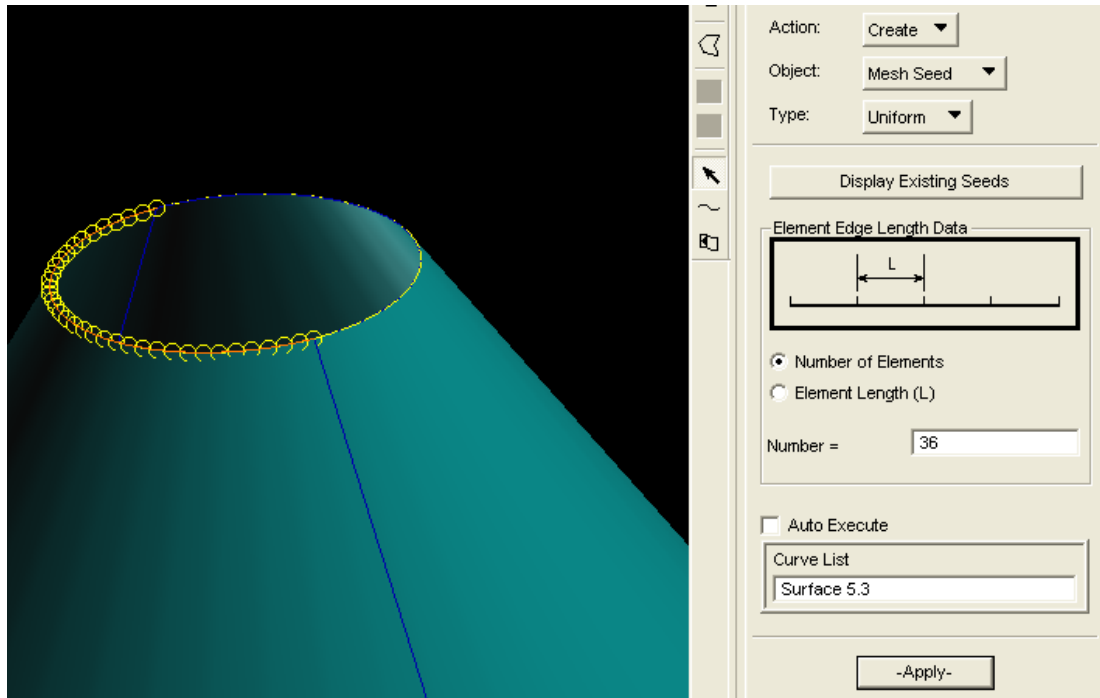
Now, we will need to complete the mesh seed around this edge surface. In the **Curve List** box, select the other portion on the same Z-axial plane. Click **Apply** to generate another 36-element mesh seed and complete the mesh seed generation on this level. The figure below shows a completed mesh seed for the top-portion of the **Cone** skin.
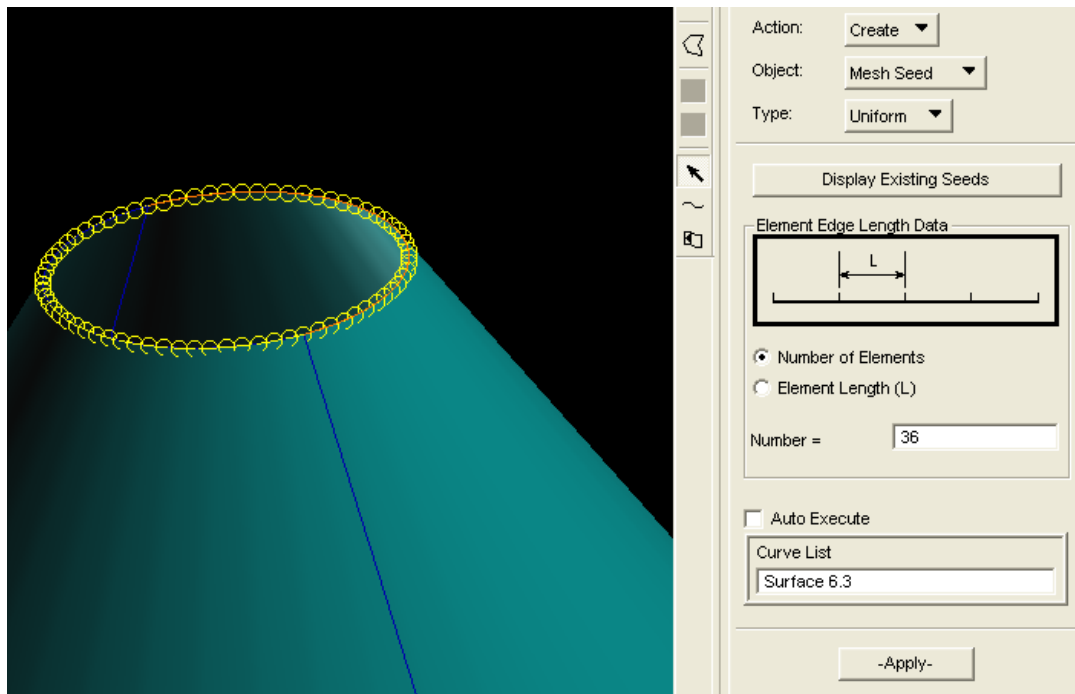
Repeat this process for all remaining horizontal-XY portions, generating 2x 36-mesh seeds for every level. Altogether, there will exist 72-mesh seeds on: the top/bottom of the **Cone**, top/bottom of the **Cylinder**, top of **Honeycomb(1)** and bottom of **Honeycomb(2)**.

The table below will guide you in correctly selecting the surface edges on each level. For every new mesh seed, input the proper surface ID into the **Curve List** entry box.

Mesh Seed	1st Surface-Half	2nd Surface-Half
CONE (Top)	Surface 5.3	Surface 6.3
CONE (Bottom)	Surface 5.1	Surface 6.1
CYLINDER (Top)	Surface 3.3	Surface 4.3
CYLINDER (Bottom)	Surface 3.1	Surface 4.1
HONEYCOMB (1) (Top)	Surface 8.3	Surface 10.3
HONEYCOMB (2) (Bottom)	Surface 7.1	Surface 9.1

Table VII – Horizontal Mesh Seed Surface Designations

Note: The surfaces designated by the above table will only be accurate if you have been following this tutorial in order. Regardless, as a few of these floating edge surfaces lie very close to each other, caution should be taken to ensure the correct edge is inputted.

An image of a fully-seeded assembly is shown in the figure below.
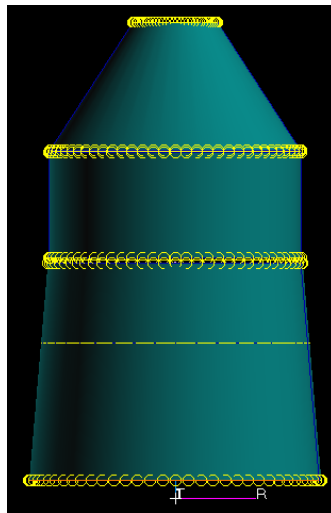
Next, we will need to create the vertical mesh-seed elements.

In the **Number** entry box, enter in the appropriate number of vertical mesh seeds based on the edge you are applying it to. This will need to be done twice for each of the four

skin layers. The specific surface ID and number of seed elements on each skin is given in the table below.

Mesh Seed	1st Surface-Half	2nd Surface-Half	# of Elements
CONE	Surface 5.2	Surface 5.4	15
CYLINDER	Surface 3.2	Surface 3.4	10
HONEYCOMB (1)	Surface 8.2	Surface 8.4	7
HONEYCOMB (2)	Surface 7.2	Surface 7.4	13

Table VIII – Vertical Mesh Seed Surface Designations

Note: As the surfaces overlap at every vertical edge, each vertical edge will have two surface designations. Selecting either edge surface ID will work correctly. However, only one set of surface IDs has been chosen to be displayed in this table to reduce confusion.

The figure below illustrates the assembly seeded for both vertical and horizontal edges.
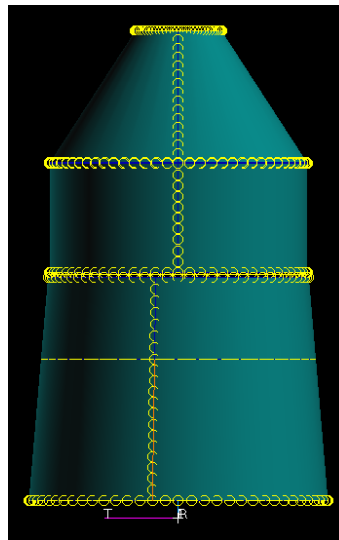
Note: The vertical mesh seeds on our model do not line up as shown in the figure above. This will not negatively impact the analysis due to the fact that these vertical mesh seeds will eventually generate mesh elements that are still equally spaced.

Now to use the mesh seed outline to generate the actual meshes across the skin.

Set **Action → Create**
Set **Object → Mesh**
Set **Type → Surface**

Set **Elem Shape → Quad**
Set **Mesher → IsoMesh**
Set **Topology → Quad4**

Ensure that the **Automatic Calculation** button has been selected in the **Global Edge Length** parameter box.

Click on the **Node Coordinate Frames** button.
Set **Analysis Coordinate Frame** and **Reference Coordinate Frame** both to **Coord 1** to set all created meshes with respect to the cylindrical coordinate frame.
Click **Ok**.

Note: Similar to how we previously generated properties for our 1D-Bar elements, we will need to generate properties for the 2D-Shell elements we are about to assign onto our QUAD4-Mesh elements.

Click **Create New Property** to bring up the properties interface.
Enter the appropriate title into the **Property Set Name** text box.
Ensure that **Object** has been set as **2D**. Also, ensure that **Type** has been set to **Shell**.

We will now create the homogeneous (forward and intermediate) skins, beginning with the **Forward_Skin_Cylinder** section. Under the **Options** selection boxes, select the appropriate option based on the respective properties of the skin being assigned as designated in the table below.

Thin, Homogeneous, Standard Formulation	Material	Material Orientation	Thickness, in	Plate Offset
Forward_Skin_Cylinder	AL7075	0	0.032	0
Intermediate_Skin_Cylinder	AL7075	0	0.200	0

Table IX – Homogeneous 2D Shell Properties

As shown for the **Forward** and **Intermediate** skins, the **Options** will be set as: **Thin**, **Homogeneous** and **Standard Formulation**.

Click **Input Properties** to open up the **2D-Shells** property settings.

For **Material Name**, click the icon and select the appropriate material type from the list based on the table above.

In the **Material Orientation** field, enter in the appropriate value and click on the **Value Type** dropdown button and select **Real Scalar** (from **CID**).

Also, enter in the proper values for **Thickness** and **Plate Offset**.

Click **Ok** to confirm property settings.
Click **Apply** to confirm property creation.

Select the **Surface List** field and use your mouse to drag-and-drop over a skin section as illustrated in the figure to the right.

For the **Forward_Skin_Cylinder** section, **Surface 5 6** should be selected or entered into the **Surface List** box designating both of these geometric surfaces.

Click **Apply** to create the first meshed surface over the **Forward_Skin_Cylinder**.
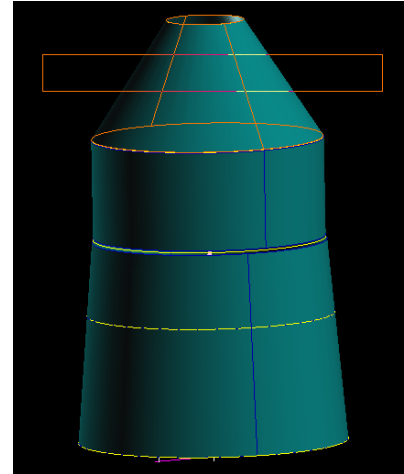
Repeat the preceding process with the **Intermediate_Skin_Cylinder** using its respective properties and surface IDs, **Surface 3 4**.

Now, in order to create the surface mesh for the Honeycomb skin we will need to slightly adjust the property configuration.

Click the **Create New Property** box.

In the **Options** selection, change the second input box to **Equivalent Section** (from **Homogeneous**) to model an equivalent composite section.
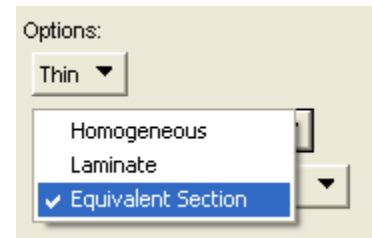
Enter a proper title for the **Honeycomb** panel in the **Property Set Name**.

Once again, click **Input Properties** to open up the property settings.
Fill in the various parameters following the information in the table below.

Thin, Equivalent Section, Standard Formulation	Membrane Material	Bending Material	Shear Material	Material Orientation	Thickness, in
Aft_Cylinder_Honeycomb	Honeycomb_Panel	Honeycomb_Panel	Honeycomb_Anisotropic	0	0.025
	Bending Stiffness	Thickness Ratio	Plate Offset	Fiber Dist. 1	Fiber Dist. 2
	333.6	10	0	0.141	-0.134

Table X – Equivalent Section 2D Shell Properties

Remember to change the **Material Orientation** dropdown parameter to **Real Scalar** (from **CID**) again.

Click **Ok** to finalize the **Honeycomb** skin's shell **Input Properties**.

Click **Apply** to confirm property creation.

In the **Surface List** field, use your mouse to drag-and-drop over both **Honeycomb(1)** and **Honeycomb(2)** sections as shown in the figure to the right. This should select the four honeycomb skin surfaces in the entry field, **Surface 7:10** (if you have been following the tutorial in order).
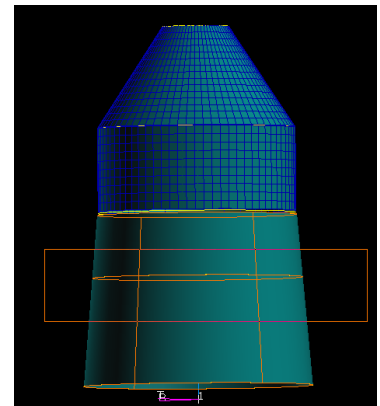
Click **Apply** to create the surface mesh across the Aft Honeycomb skin section.

Successfully surface meshing the entire thrust tube assembly should result in a model similar to the figure below.
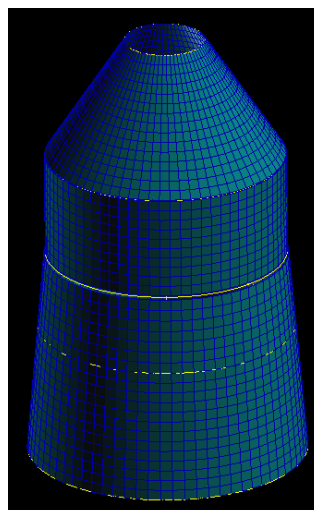
10. **Equivalencing the Model (First-Time)**

As indicated earlier, the **Bar/Quad** elements creation has generated a large number of redundant nodes in our model. These accumulated redundancies can be quickly resolved by using the **Equivalence** function in Patran.

Select the **Elements** Application Form from the Patran Main form.
Set **Action → Equivalence**
Set **Object → All**
Set **Method → Tolerance Cube**

Click **Apply** to equivalence the entire model. Several hundred nodes should be deleted.

$# Database realignment completed.
$# Geometric equivalencing completed.
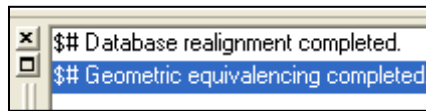
Note: When you equivalence using these settings, you do not need to select anything on the model as the entire model is equivalenced.

FEA Modeling – Tying the Model Together Using MPCs

11. **Integrating the Spacecraft Structure using Multi-Point Constraints**
    We will now begin utilizing a new type of element in Patran called the multi-point constraint or **MPC**. A wealth of information is publicly available regarding the different types of **MPC**s and their various nuances. However, for the purpose of this project, all the user needs to know is that we will be utilizing the **RBE2**-type of **MPC** to tie grids together for two primary purposes:

    a) Tie the furthest forward/aft sections at a point about the center of rotation of the spacecraft. This will allow for a single load or boundary-condition to be placed at a single point rather than splitting into multiple conditions across the outer edges of the cylinder.

    b) Assemble and tie the floating **Intermediate** and **AKM** rings with their neighboring skin sections. This will allow for loads/displacements to be transferred across the assembly structure despite the geometry/nodes not being physically connected.

    *Note: **RBE2 MPC**s have been chosen as they fulfill our requirements. There will be one independent grid-node (with all six degrees-of-freedom) and may have one or more dependent grid-nodes (with a set number of degrees-of-freedom). The **RBE2** allows us to rigidly tie, or "weld", multiple nodes to one other grid point.[4]*

12. **Applying Multi-Point Constraints (MPCs) to the Model, Part A**
    We will begin with the first type: creating a "spider-web" **MPC**-element that will allow us to place a single **Load** or **Boundary-Condition** along the axis-of-rotation of the spacecraft. To do this, we will create and place two nodes at the center of the actual physical location of the **Forward Ring** and the **Separation Ring**.

    Select the **Geometry** Application Form from the Patran Main form.
    Set **Action → Create**
    Set **Object → Point**
    Set **Method → XYZ**

    In **Reference Coordinate Frame**, enter or type in **Coord 1** to select the cylindrical coordinate system.

    In **Point Coordinates List**, enter the central location of a ring. This will be **[0 0 Z-location]** where the Z-axial location is with respect to which **Ring** you are generating, based on the table below.

	Z-location
Forward Ring	60.272606
Separation Ring	1.1925808

Table XI – Cylindrical Z-axis Coordinate Location of Rings

Click **Apply** to generate the **Point**.

Select the **Elements** Application Form from the Patran Main form.
Set **Action → Create**
Set **Object → Node**
Set **Method → Edit**

In **Analysis Coordinate Frame** and **Coordinate Frame**, set to **Coord 1** to utilize the cylindrical coordinate frame for the node we will create.

In the **Node Location List**, enter or select the **Point** we just created.

Click **Apply** to generate the **Node**.

*Note: Now, we will create the **MPC** tying the newly created floating **Ring Node** to the **Nodes** surrounding the top or bottom skin structures.*

Set **Action → Create**
Set **Object → MPC**
Set **Type → RBE2**

Select the **Define Terms…** button to open up the **MPC Node** definition box.

Note: For tying the two RBE2 nodes together, the choice of independent and dependent node is not relevant. However, for the sake of consistency, the independent node will be the floating ring node and the dependent nodes will be the adjacent skin node.

Select the **Create Independent** button.
In the **Node List** selection box, select the **Node** you have just created (that represents the center of the true physical location of the ring).
Click **Apply** to add this node onto the **Independent Terms** list.

Select the **Create Dependent** button.

Ensure that all six degrees-of-freedoms (**UX**, **UY**, **UZ**, **RX**, **RY**, **RZ**) are selected in the **DOFs** selection box.

In the **Node List** selection box, type in the **Node** IDs for all the nodes on the skin adjacent to the floating node. This will be all 72 nodes surrounding either the [**Forward Ring Bar-**element/**Forward Cone**] or the [**Separation Ring Bar**-element/**Honeycomb (2)**]. You may need to use the **Action: Show**, **Object: Node**, **Info: Location** function to find the exact IDs of these **Nodes**. Any node IDs shown in the format x:y:z (i.e. 2:142:2 in the example on the figure below) is interpreted as the **Node** IDs from x to y, counted in intervals of z.
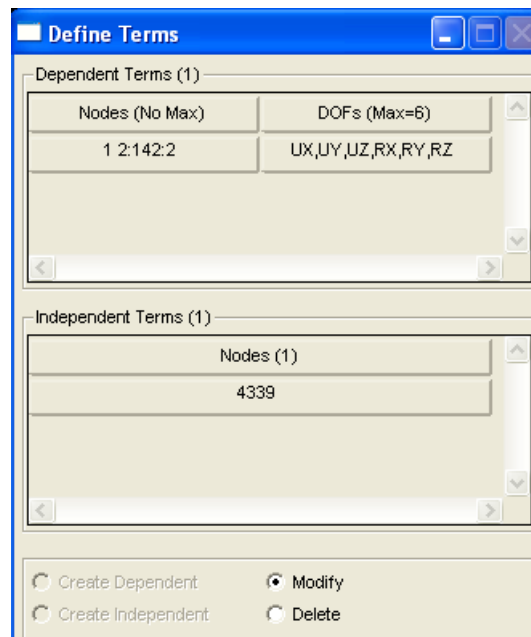
*Note: At this point, after equivalencing and deleting various nodes, the exact list of **Node** IDs may not be immediately intuitive. By creating the **Bar**-elements using the **Transform** method, the second point in a bar, n, also has the first point in the next bar, n+1, that overlaps its location. This is the reason why, after equivalencing, the **Node** IDs will no longer be in simple sequential order (i.e. **Nodes 1:72**). Instead, the **Node** IDs will be in a range greater than 72 and in intervals of greater than 1 (the example in the figure above shows IDs from 2 to 142 in intervals of 2 or 2,4,6….142). However, you'll notice Node ID 1 and 2:142:2 still come out to exactly 72 **Nodes**.*

Inputting the proper ID list into this parameter is imperative.

Click **Apply** to add these 72 nodes onto the **Dependent Terms** list.

Click **Cancel** to close the term definition box.

Click **Apply** to finalize creation of the **MPC**.
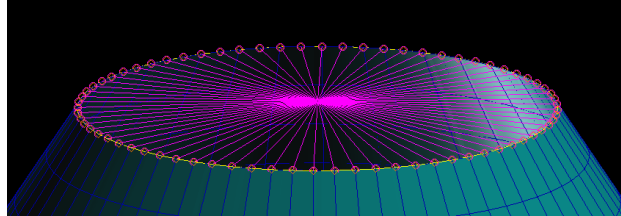
Successfully generating this **MPC** should result in a "Spider-Web" looking, purple-colored element.

If you have generated the forward-**MPC**, you will now need to repeat the process for the aft-**MPC** and vice versa. After generating both sets of **MPCs**, the model should look similar to the figure below.

*Note: Now the model can have a **Load** or **Boundary-Condition** placed at the central **Independent Node** and distribute it over the tied **Dependent Nodes** on the outer cylinder edges of the assembly skin.*
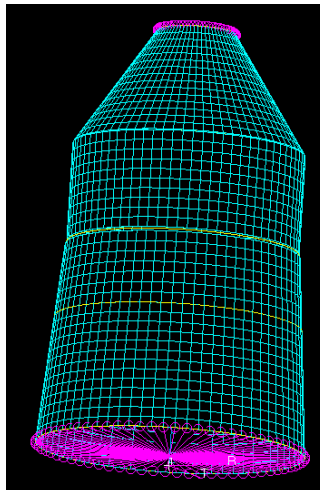
13. **Applying Multi-Point Constraints (MPCs) to the Model, Part B**
    Now we will create **MPC**s that integrate the skin structures together so that a load will be seen contiguously through the entire spacecraft assembly.

    Select the **Elements** Application Form from the Patran Main form.
    Set **Action → Create**
    Set **Object → MPC**
    Set **Type → RBE2**

    Select the **Define Terms…** button to open up the **MPC Node** definition box. For tying the floating ring with the two nearby skins, the independent node will be the ring node and the dependent nodes will be the two adjacent skin nodes.

    Select the **Create Independent** button.
    In the **Node List** selection box, select a **Node** from one of the two floating rings, **Intermediate** and **AKM**, not tied to any skin nodes.
    Click **Apply** to add this node onto the **Independent Terms** list.

    Select the **Create Dependent** button.
    Ensure that all six degrees-of-freedoms (**UX**, **UY**, **UZ**, **RX**, **RY**, **RZ**) are selected in the **DOFs** selection box.
    In the **Node List** selection box, either type in or select (using Shift+Left-Mouse-Button) two adjacent skin **Nodes** to the **Independent Node** just selected: one from a skin structure above and one from a skin structure below.
    Click **Apply** to add these two nodes onto the **Dependent Terms** list.
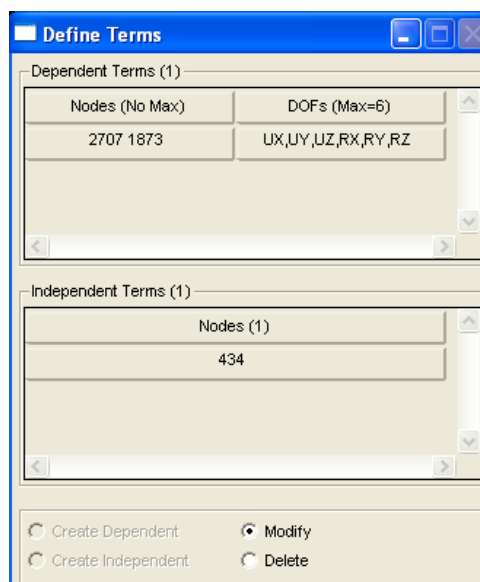
Select **Cancel** to close the term definition box.

Click **Apply** to finalize creation of this **MPC**.
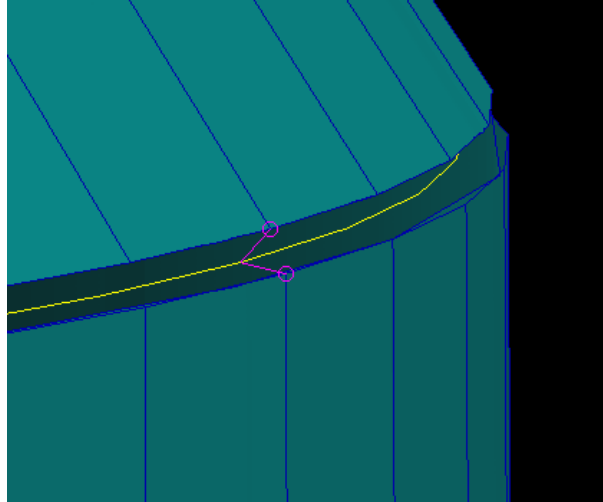
*Note: You will notice that we cannot use the **Transform** function to rotate-and-clone the **MPC**-elements around the cylinder. However, there is another function in Patran that allows us to create a **Group** out of any type of entity whether that is a **Point**, a **Node**, an **Element**, an **MPC**, etc. Using a created **Group**, we may **Transform** and clone this entity with all its properties intact.*

On the Main Menu bar, click **Group → Create…**
In **New Group Name**, give the **MPC** an appropriate title.
Click in the **Entity Selection** box and select or enter the previously created **MPC**. This should be **Mpc 3** for the first **MPC** created in this section of the tutorial (if you have been following this tutorial in order).
Click **Apply** to finalize the creation of this **Group**.

*Note: Now to **Transform** and rotate this single-**MPC Group** 71 times.*

On the Main Menu bar, click **Group → Transform…**
Set **Action → Transform**
Set **Method → Rotate**

Ensure that the appropriate ring **MPC** is selected in **Selected Group(s)**.

We will be copying the **MPC** every five-degrees, seventy-one times around the cylindrical axis to create an **MPC** at every node on the **Ring BAR** elements.

54

Select the **Copy** button as we will be rotating and copying this **RBE2 MPC**.
In the number of **Time(s)** entry box, type in **71**.

In **Reference Coordinate Frame**, type **Coord 1** to reference the transformation to the cylindrical axis we created earlier.
In **Axis**, type **Coord 1.2**. This will allow for proper Z-axis node rotation based on our current coordinate system settings.

In **Rotation Parameters**, we will transform and copy this **MPC** every five-degrees. In **Rotation Angle**, set to **5**.

Click **Apply** to finalize the **MPC** transform process.
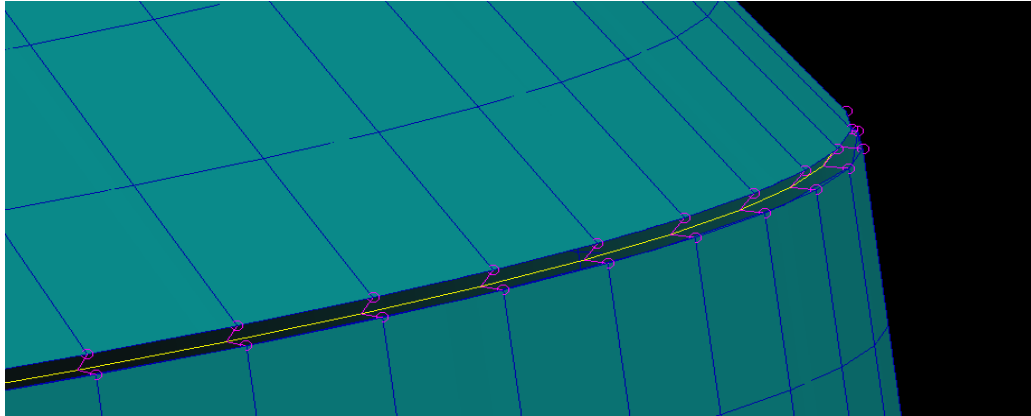
If you have tied the **Skin** and **Ring Elements** with the MPC for the **Intermediate Ring**, you will now need to do this same procedure for the **AKM Ring**, or vice-versa.

Completing this process, should result in a total of 146 **MPC**s that tie all the **Skin** and **Ring Elements** together. A correctly tied thrust tube assembly model is shown in the figure to the right.

The **RCS Ring** is already directly integrated with nodes on the skin and so it does not need a tied **MPC** as with the other **Ring Elements**.

The thrust tube assembly model is now completed and ready for analysis.
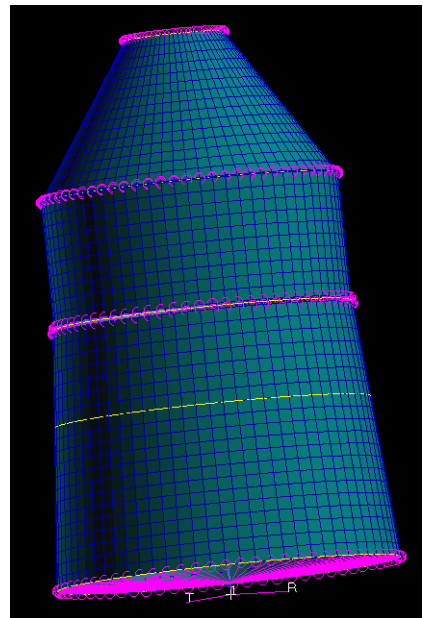
14. **Equivalencing the Model (Second-Time)**

Similar to the previous time we used the equivalence function to rid redundant nodes after **Bar/Quad** element creation, we will need to utilize the function again after **MPC** cloning. The **Group Transform** for creating our **MPC**s has also generated a large number of redundant nodes in our model which will again be eliminated using the **Equivalence** function.

Select the **Elements** Application Form from the Patran Main form.
Set **Action → Equivalence**
Set **Object → All**
Set **Method → Tolerance Cube**

Click **Apply** to equivalence the entire model. Several hundred nodes should be deleted.
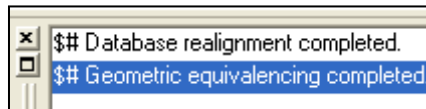
FEA Modeling – Setting up the Analysis Conditions

15. **Setting up the Test Analysis Loads and Boundary Conditions**
We will now create a set of unit-load test cases in order to ensure that the model has been created properly. A single force or moment on each axis provides a simple test case to ensure that the model is operating appropriately. Any incorrectly floating nodes or improperly modeled **MPC**s will become immediately apparent as the model will twist or bend in a strange manner.

We will be creating 6 **Load Cases**: one unit-load case for each axis for both force and moment. Each of these unit-load cases will consist of the same **Displacement Boundary Condition** and six different 1000lb or 1000in-lb **Force/Moment Load Conditions**.

The procedure for setting this up properly is as follows:

> 1) Create the individual forces, moments or displacement boundary conditions using the **Loads/BCs** Application Form

> 2) Create a test case scenario using multiple loads and/or boundary conditions using the **Load Case** Application Form

The six different **Force/Moment Load Conditions** are shown in the table below.

	1000LBS Unit-Force Loads					
	X_Force	**Y_Force**	**Z_Force**	**X_Moment**	**Y_Moment**	**Z_Moment**
Force	$< 1000, 0, 0 >$	$< 0, 1000, 0 >$	$< 0, 0, 1000 >$	$< 0, 0, 0 >$	$< 0, 0, 0 >$	$< 0, 0, 0 >$
Moment	$< 0, 0, 0 >$	$< 0, 0, 0 >$	$< 0, 0, 0 >$	$< 1000, 0, 0 >$	$< 0, 1000, 0 >$	$< 0, 0, 1000 >$
Analysis Coord Frame	Coord 0	Coord 0	Coord 0	Coord 0	Coord 0	Coord 0

Table XII – Individual Unit-Force Loads and Moments

Select the **Loads/BCs** Application Form from the Patran Main form list.
Set **Action → Create**
Set **Object → Force**
Set **Type → Nodal**

In the **New Set Name**, enter an appropriate label for the specific load. (i.e. XForce for the 1000lb load on the X-axis)

Select the **Input Data** button to enter in the appropriate data from the table above specific to the load case you are setting up. Fill in values for **Force**, **Moment** and **Analysis Coordinate Frame**.

Click **Ok** to finalize the inputs.

Click on the **Select Application Region** to choose where the unit-load will be placed relative to the assembly structure.

On the top of the screen, enter the **Select** scroll-down menu to choose which type of property will be allowable for application region selection. Select **FEM** from the selection choices.

ALL loads will be located at the center of the **MPC** on the **Forward Ring**. Either select this node manually or enter it in to the **Select Nodes** field. This should be **Node 4339** (if you have been following this tutorial in order).

Click **Add** to add the node into the application region list.

Click **Ok**.

Click **Apply** to finalize the first load creation.
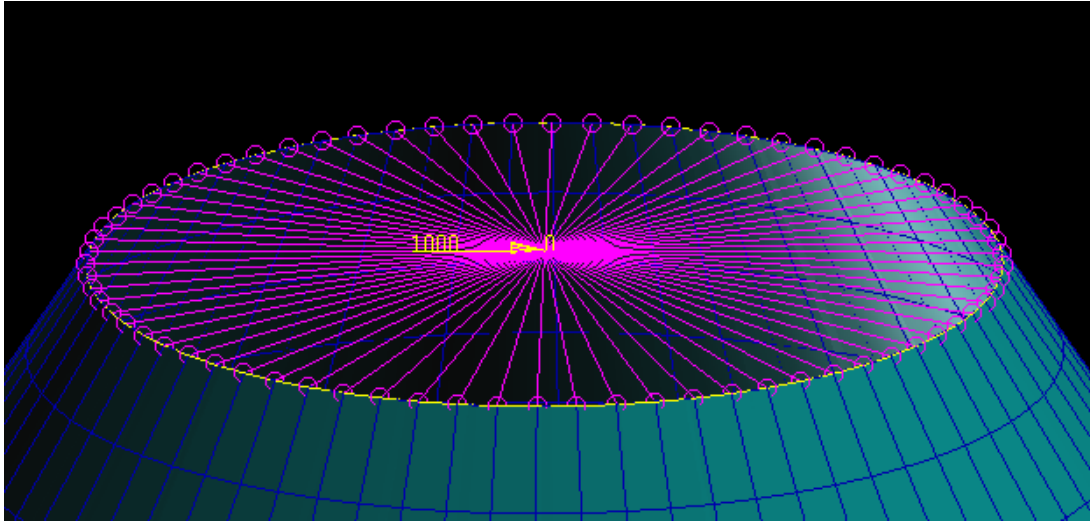
Repeat this process for the other five loads. Keep in mind that the displayed forces/moments on the screen will begin to look strange as soon as you have created two **Loads/BCs**. This is because as you are creating each load into a **Default Load Case**, they will display as a sum of all combined **Loads**. Don't worry about this as we will be separating these loads into specific **Load Cases** so that analysis will be performed one load at a time.

The **Displacement Boundary Condition** will remain the same for all six **Load Cases** is shown in the table below.

	Displacement BC
Translations	< 0, 0, 0 >
Rotations	< 0, 0, 0 >
Analysis Coord Frame	Coord 1

Table XIII – Displacement Boundary Condition

Select the **Loads/BCs** Application Form from the Patran Main form list.
Set **Action → Create**
Set **Object → Displacement**
Set **Type → Nodal**

In the **New Set Name**, enter an appropriate label for the boundary condition we are creating. (i.e. UnitLoad_BC)

Select the **Input Data** button to enter in the appropriate data from the table above for the

boundary condition we are setting up. Fill in values for **Translations**, **Rotations** and **Analysis Coordinate Frame**.
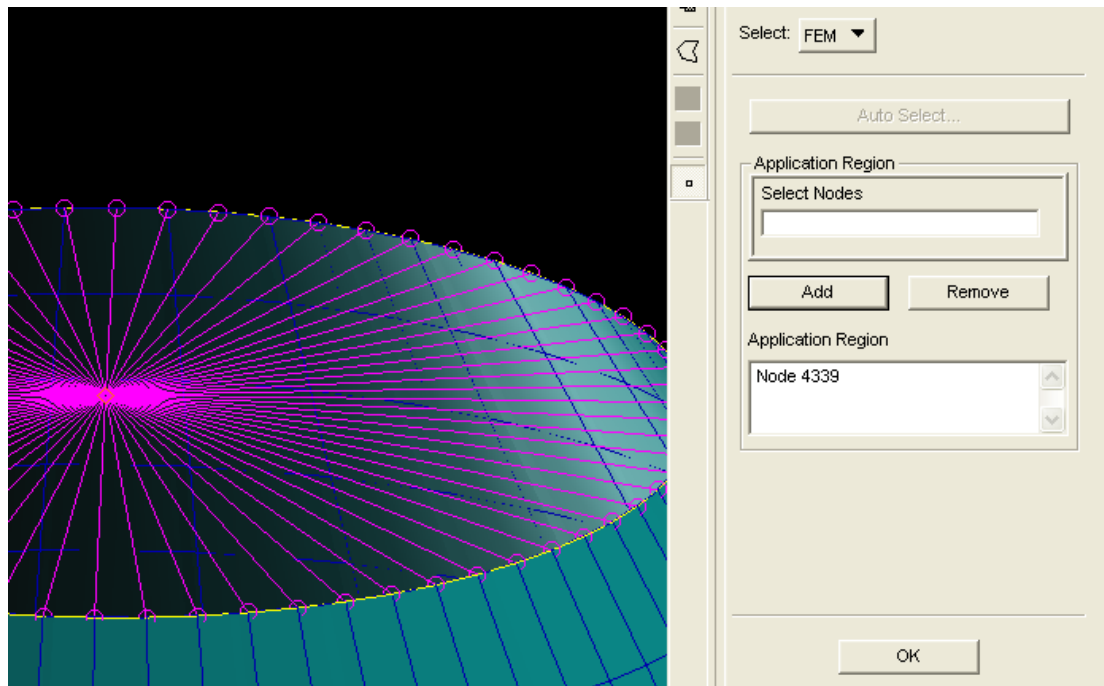
Click **Ok** to finalize the inputs.

Click on the **Select Application Region** to choose where the boundary condition will be placed relative to the assembly structure.

On the top of the screen, enter the **Select** scroll-down menu to choose which type of property will be allowable for application region selection. Select **FEM** from the selection choices.

The boundary condition will be located at the center of the **MPC** on the **Separation Ring**. Either select this node manually or enter it in the **Select Nodes** field. This should be **Node 4340** (if you have been following this tutorial in order).

Click **Add** to add the node into the application region list.

Click **Ok**.

Click **Apply** to finalize creation of our test boundary condition.
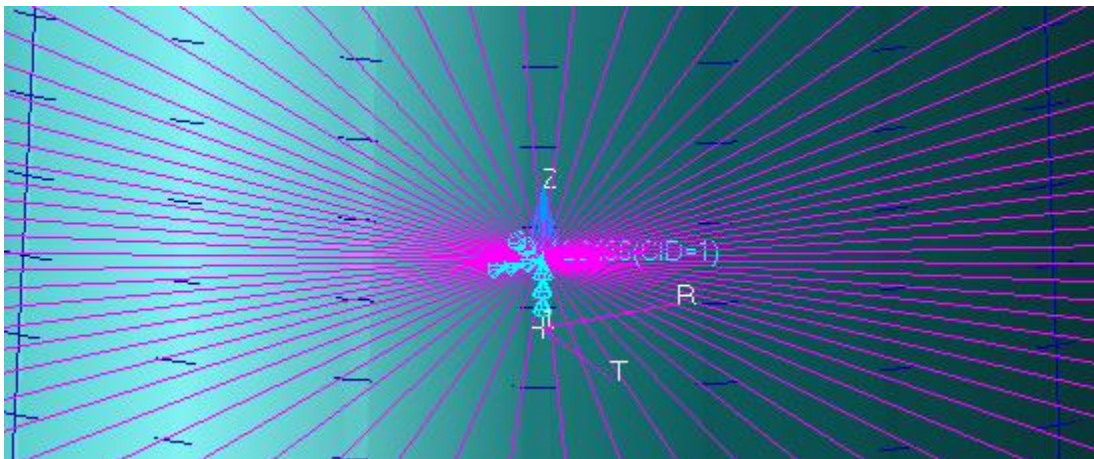
16. **Creating the Load Cases for Model Analysis**

    *Note: Now we can begin using the loads and boundary conditions we have just created and integrating them together to form a **Load Case** for NASTRAN analysis.*

    Select the **Load Cases** Application Form from the Patran Main form list.
    Set **Action → Create**
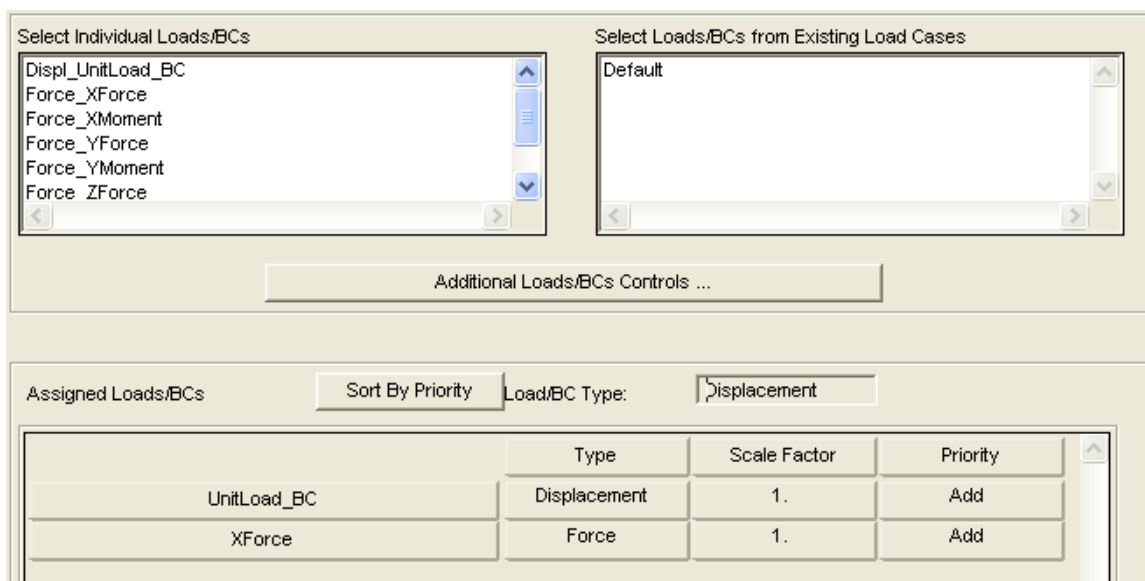
    Enter an appropriate for the **Load Case Name** based on the type and direction of load you are about to set. (i.e. **XForce_1000lbs**).

    Click the **Input Data** button to select which loads and boundary conditions will be utilized by the load case.

    In the **Select Individual Loads/BCs** selection area, choose one load and the boundary condition for one of the six different unit loads to be tested. After being selected, the load or boundary condition will appear on the **Assigned Loads/BCs** area at the bottom of the **Input Data** screen.

    *Note: The labels of all loads and boundary conditions will be in the format Displ_[YourLabel] for displacement boundary conditions and Force_[YourLabel] for loads. For every load case, there should only be two items placed in two rows of the spreadsheet. If you accidentally select the wrong load, click that load on the spreadsheet and click the button at the bottom titled **Remove Selected Rows**.*

    Click **Ok** once you have finished load and boundary condition assignments for your first **Load Case**.

Click **Apply** at the bottom to finalize **Load Case** creation. In the **Existing Load Cases** list at the top, your new **Load Case** should be appended onto the list following the **Default** case.

Repeat this process for all six unit-load test cases.

This can be done by Chenging the **Load Case Name** to a new and appropriate title. Then, go into **Input Data** and click the only load and **Remove the Selected Rows**. Finally, click on the new load you wish to create. Click **Ok** and then click **Apply**.

When this is successfully completed, you will have six **Load Cases** corresponding to a unit-load in each axis: 3 Forces and 3 Moments.

We can now proceed to using these test cases for analysis to physically check the validity of our model.
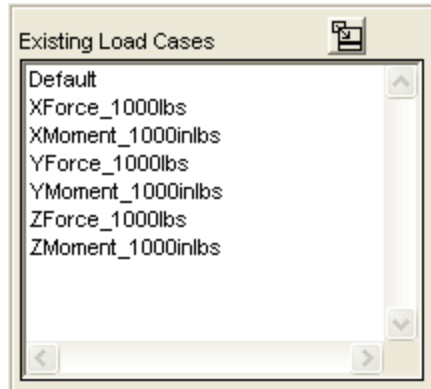
FEA Modeling – Running the Analysis and Accessing Results

17. **Running the Analysis using PATRAN and NASTRAN**
    Now we will simulate these load cases on our thrust tube assembly model.

    Select the **Analysis** Application Form from the Patran Main form list.
    Set **Action → Analyze**
    Set **Object → Entire Model**
    Set **Method → Full Run**

    Click on the **Subcase Select…** button at the bottom of the selection panel on the right. All the different load cases you have just created should be available for selection.

    Deselect the **Default** case from the **Subcase Selected** list on the bottom and select all of the six different load cases so that they appear in the **Subcase Selected** list.

    Note: Although all the load cases have been chosen, Nastran will later interpret these loads and boundary conditions individually. The forces and moments will not be combined into one "case". Results can be viewed for each unit-load separately so do not worry that all of these cases have been chosen.

    Click **OK** to finalize the load case selections.
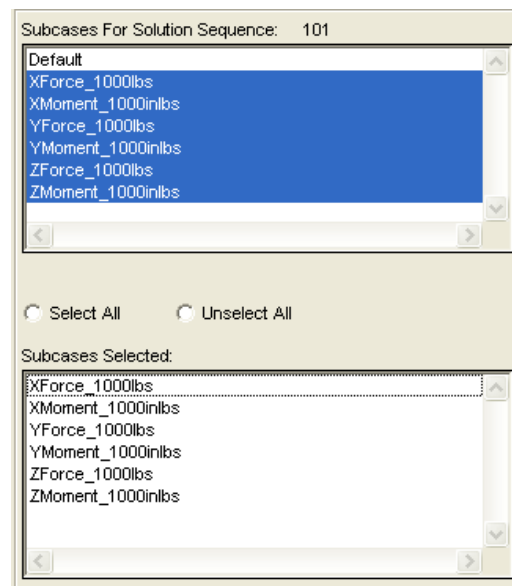
    In the **Job Name** entry box, enter an appropriate title that will be easy for you to keep track of.

    Click **Apply** to begin analysis. This will take several seconds to several minutes depending on the computer resources available.

```
 MD Nastran V2008.0 (writeuptest)                                   _ □ ×
    IterSolver    =
    NumMachinesCmd =
    CommandInputs  =
    WaitFlag      = 0
Job submitted to local host with:
c:\MSC.Software\MD_Nastran\md2008\i386\nastran.exe WriteUpTest.bdf    bat=no

*** USER WARNING MESSAGE (pgm: nastran, fn: decode_keyword)
    batch=no    (command line[2])

    This keyword is not available on this platform.
MD Nastran V2008.0 (Intel Windows XP 5.1 (Build 2600)) Fri Oct 15 20:35:08 2010

*** SYSTEM INFORMATION MESSAGE (pgm: nastran, fn: estimate_job_requirements)
    Starting ESTIMATE, please wait...

*** USER INFORMATION MESSAGE (pgm: nastran, fn: estimate_job_requirements)
    Estimated DOF=16062
    Estimated memory=128.0MB
    Estimated disk=78.4MB

    Note: ESTIMATE had some difficulty analyzing this job.
MD Nastran beginning job writeuptest.
MD Nastran started  Fri Oct 15 20:35:08 PDT 2010
```

Do not shut down or disturb the computer until all results are processed the MS-DOS prompt box (shown in the figure above) has closed itself.

Note: Several output files using this title name will be created once Patran has sent this FEA job to Nastran. These files will be created onto the desktop and contain the bulk

data file (.bdf), the F06 file (.f06) as well as several other files that are not as important. The F06 file can also be accessed using the **Action → Monitor** *and* **Object → Job**. *Access to the file will be then listed in the selection buttons at the bottom.*
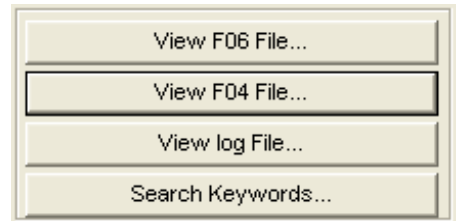
The bulk data file is the completed node, mesh, material properties and load case information for all information you have in your Patran model. This is the crux of what Nastran takes from the Patran interface to do any FEA analysis. Resources are available in Patran reference manuals that explain the contents of the bulk data files in detail. These can be obtained through contact with David Esposto (805-756-5136, desposto@calpoly.edu).

The F06 file provides a summary of the analysis results and should be a very long text file containing analysis separated based on a labeled subcase. This file is useful in that it can tell you if something in your model was done incorrectly as a fatal error in Nastran will result in a very short F06 file that details what the error entails giving you some idea as to where and how to fix it. Always check this file after any analysis is performed to make sure all subcases were analyzed to completion.

18. **Accessing Nastran Results**

After Nastran has finished analyzing all six load cases, we will need to access the results file. Though we could directly access the results text file, by using Patran to view the results file we can have useful visualizations and animations that will help us understand these results. The unit-load tests we have used for this tutorial are merely a simulation to assess the validity of our spacecraft assembly model. Checking these results using the visualization options we have available from Patran will tell us immediately if the **Mesh** and **Bar** elements have not been hooked up properly.

Select the **Analysis** Application Form from the Patran Main form list.
Set **Action → Access Results**
Set **Object → Attach XDB**
Set **Method → Result Entities**

Click on the **Select Results File** and select the results file (.xdb) generated from Nastran. This will be in the format *[Your_Job_Name].xdb*.
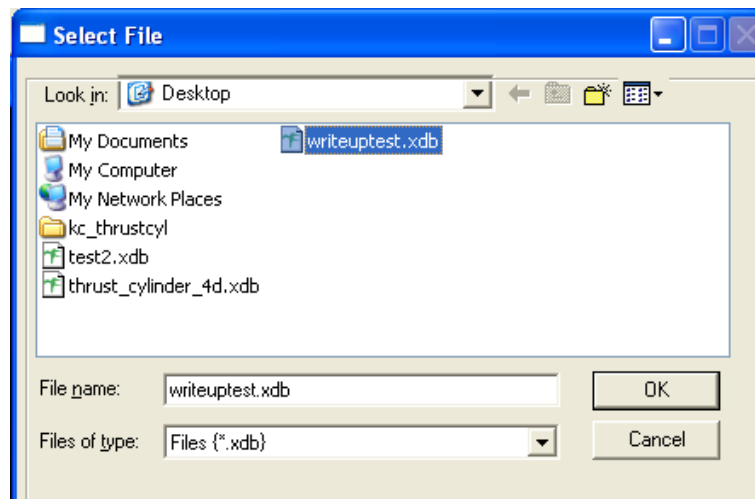
Click **Ok** to close this menu.

Then, click **Apply** on the bottom of the screen to attach the interested results file into Patran memory for access.

19. **FEA Model Results**

Now we will use the various options available in Patran to view the results of the different load cases on our model.

Select the **Results** Application Form from the Patran Main form list.
Set **Action → Create**
Set **Object → Quick Plot**

In the **Select Result Cases**, choose a subcase you are interested in viewing the results for.

Then, select any **Fringe** and/or **Deformation** result. For example, choose **Displacements, Translational** for both **Select Fringe Result** and **Select Deformation Result**.

Either de-select or leave the **Animate** option at the bottom active.

Click **Apply** to view the result for the unit-load you have selected.

*Note: To delete these graphics off your model, Set **Action → Create** and **Object → Quick Plot**. Select and highlight all items in the **Existing Plot Types**. Then, click the **Apply** button.*

Tables

Table I – Spacecraft Relevant Material Properties

Table II – Spacecraft Ring Method (A) Properties

Table III – Spacecraft Ring Method (A) Offset Locations

Table IV –Spacecraft Ring Method (B) Offset Locations

Table V – Spacecraft Ring Method (B) Properties

Table VI – Ring-Specific Element IDs

Table VII – Horizontal Mesh Seed Surface Designations

Table VIII – Vertical Mesh Seed Surface Designations

Table IX – Homogeneous 2D Shell Properties

Table X – Equivalent Section 2D Shell Properties

Table XI – Cylindrical Z-axis Coordinate Location of Rings

Table XII – Individual Unit-Force Loads and Moments

Table XIII – Displacement Boundary Condition

References

[1] Patran User's Guide, pp 12.
[http://simcompanion.mscsoftware.com/infocenter/index?page=content&id=DOC9203&cat=PATRAN_DOCUMENTATION_2008_R1&actp=LIST]

[2] http://step.nasa.gov/help/faq.html

[3] http://asm.matweb.com

[4] "MPCs in Nastran.pdf", pp 12-17. Email from Ray Ng, "RE: InterSEP FEA modeling help", Sept 14, 2010.

VI. FEA Results & Analysis [Jason Carpenter]

Test Cases

In order to validate our FEA Model, six test load cases were implemented. We applied three axial loads and three moment loads of 1,000 lbs and 1,000 in-lbs respectively. By running these load cases, we are able to check proper connections between the skin sections and verify that all of the material and physical properties are modeled correctly. In order to perform the hand calculations, we had to alter the simplifying assumptions some. The rings are removed from the structure and it is assumed that the center skin stretches to cover the vertical distance that was occupied by the rings.

Load Case Setup

For all of the test cases, the loads and constraints were applied at the same locations. The figure on the next page, shows the locations for the applied forces and constraints. To be congruent with its attachment to the rocket, we constrained the structure at the separation ring. The ring is constrained in the radial and y (vertical) directions, denoted by the blue double headed arrows in parts a and c of the figure on the next page. The ring is not constrained in the theta (rotational) direction since the spacecraft is constrained with a clamp band, which allows for rotation about the y-axis. Multi-Point Constraints (MPC), shown in pink, are added to the top and separation rings to allow for the constraints/loads to be applied evenly to the entire ring. The constraints were applied to the central hub of the MPC on the separation ring. At the top ring, the selected load is applied to the central hub of the MPC as well, shown as the yellow arrow in parts a and c of the figure on the next page.

Results Cases

In testing the results of the finite element analysis, we performed simplified hand calculations to compare the solutions with. Since the two methods require different assumptions we do not need answers that are exactly the same to validate the model. We looked for answers that are close, i.e. within an order of magnitude. In addition, we looked at the difference in results and made sure there are reasonable explanations for the incongruence. Since we cannot calculate the displacements at every point by hand due to the volume of calculations required, we examined only a select number of points.
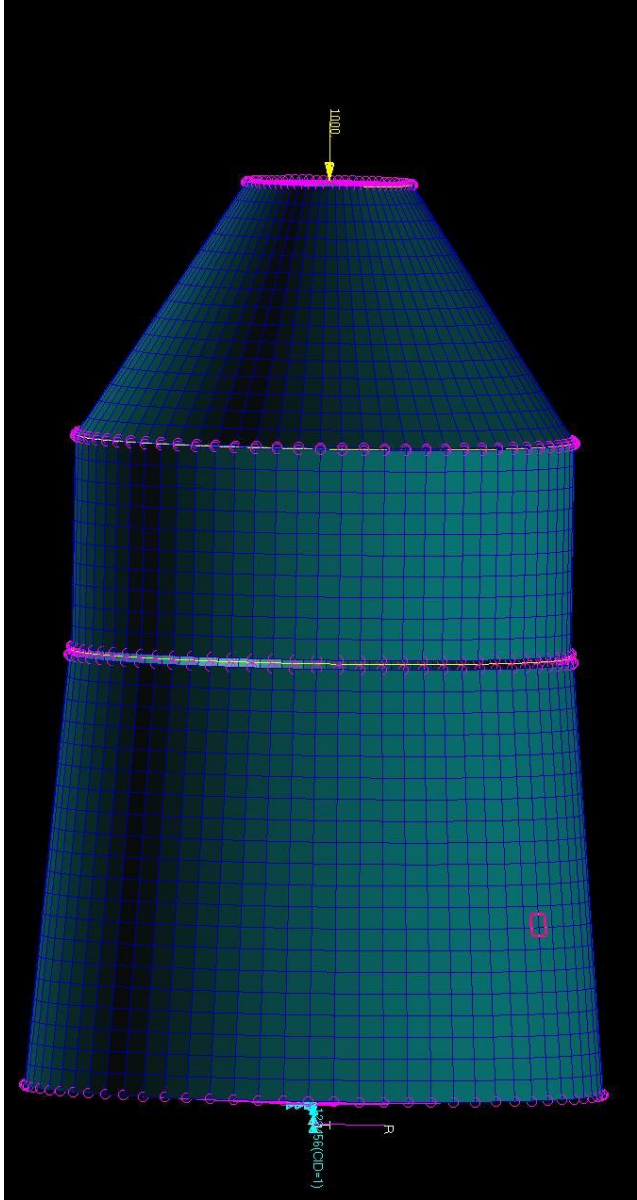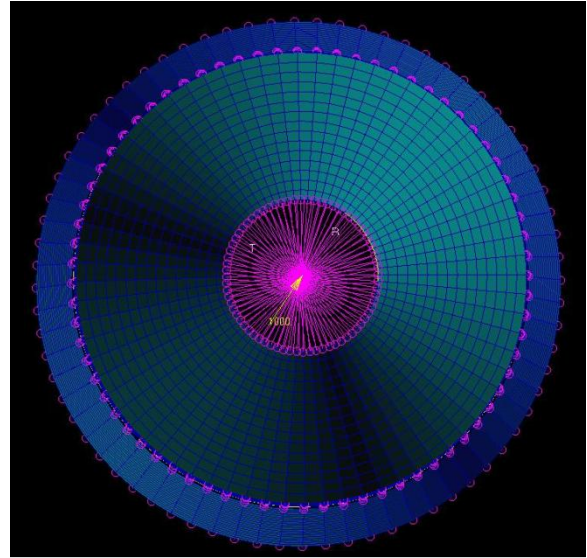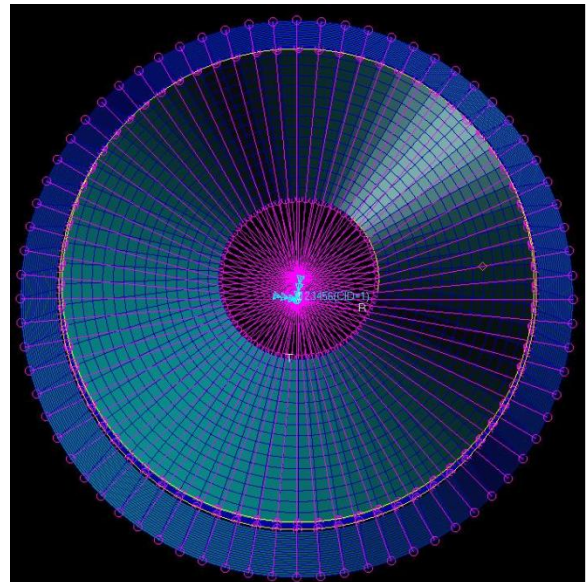
a) Side View   b) Bottom View
Locations of the applied loads and constraints.

Y-Axial Case

The first case that was examined was a 1000 lb axial load in the negative y direction. This simulates the weight force of the spun section and other components on the thrust cylinder. The stress at each y location of the structure was calculated using the formula:

$$ - \tag{1}$$

By calculating the cross sectional area for each y location in the cylinder, the stress at each location was calculated using MATLAB. Using Hooke's law:

$$\tag{2}$$

the local strain for each y location was calculated. The displacement was then found by multiplying the strain by a differential height and numerically integrating over the entire height of the cylinder; the results are shown in the top figure on the next page. The displacement ranges from zero at the separation ring and has a maximum displacement of 1.73E-3 inches at the uppermost edge of the forward skin. Since the area varies linearly in the two conical sections, the stress distribution and subsequently the displacements varies as the inverse of y.

When looking at the finite element analysis, we find similar results.   The FEA output is shown below in the second figure on the next page. Looking at it we see the same trends for the displacement. In the figure, the displacement has been discretized into 12 steps represented by the different colors. The non-linear nature of the displacement can be seen by the changing width of the color bands on the model itself. As the bands get narrower, the displacement is changing more rapidly indicating a change in the slope as seen in the plot. Perhaps the best indicator is the maximum displacement. From the FEA result, it can be seen that the maximum displacement given by FEA is 1.95E-3 inches, which is only an 11% difference. That shows that these two models do correlate very well.
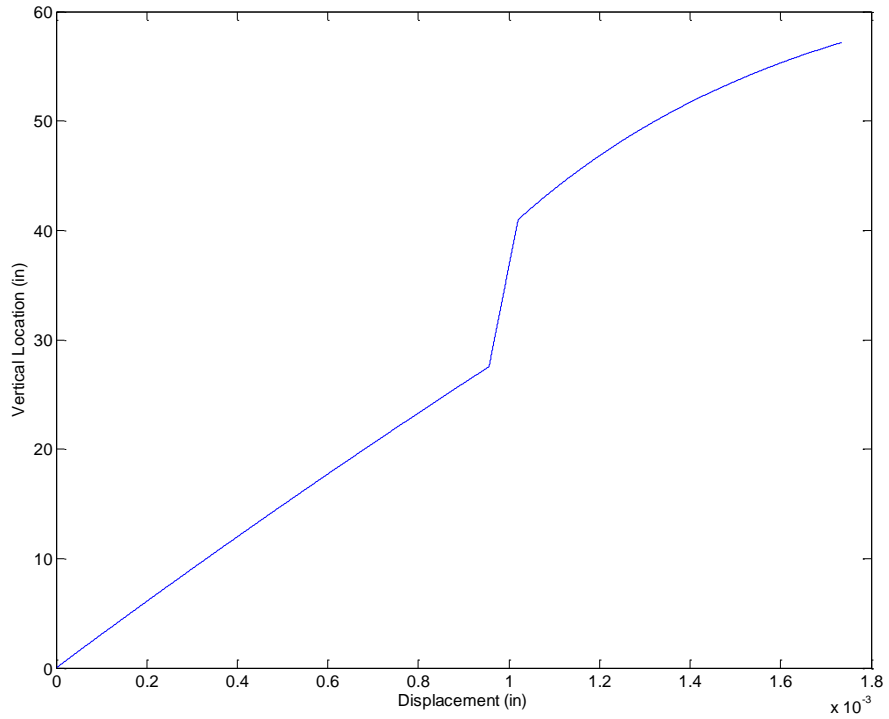
Simplified Calculation of Displacement as a function of Y location
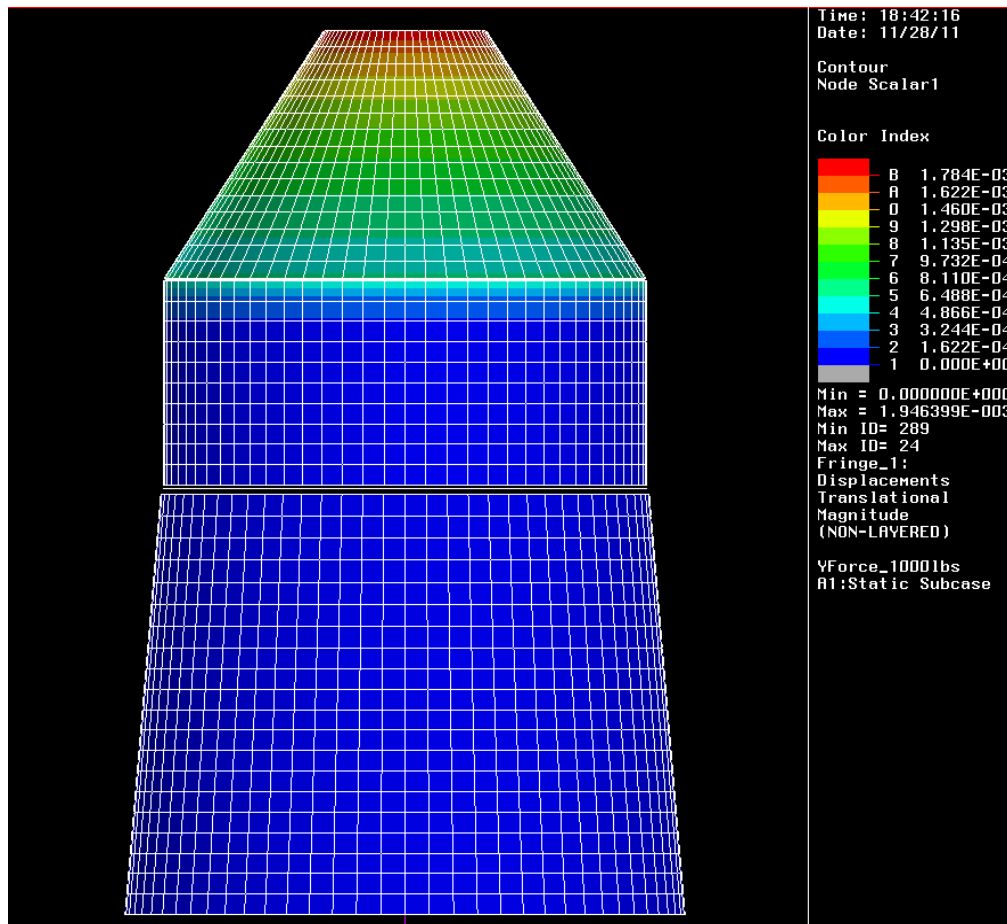
FEA Results for displacement

X,Z Axial Load Case

The next two load cases applied can actually be combined together due to symmetry for the interest of convienience. Since our cylinder is completely symmetrical about the central axis, the X and Z axial load cases are simply rotations of each other. As such, the results will be the same just shifted about the y axis by 90 degrees. In order to simpify the structure for hand calculation analysis, the cylinder was modeled as having a single diameter and thickness, shown in the figure below, resulting in a constant I (moment area), A (cross sectional area), and E (young's modulus).
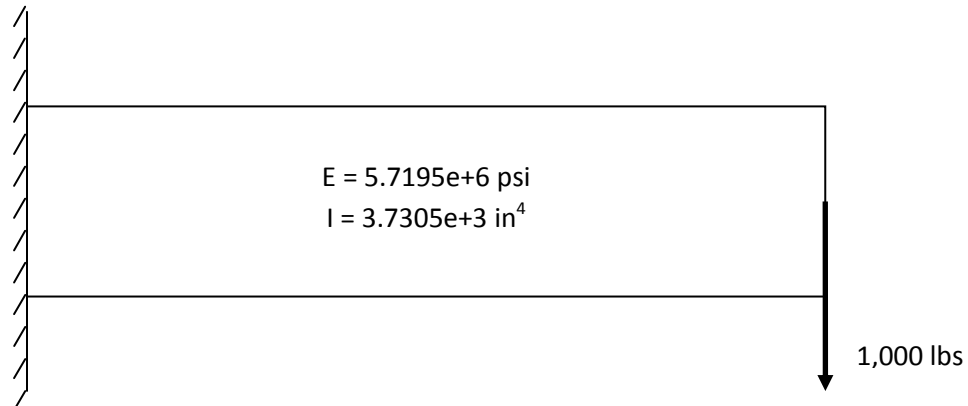
Engineering Model for Hand Calculations of X and Z Axial Load Cases

To get the highest fidelity, a weighted average was taken for E and I resulting in the values shown in engineering model figure above. By simplifying the model to this case, we can use the displacement equation for a simple cantilevered beam. Using the given equation:

$$\longrightarrow \tag{3}$$

the displacement for each y value was calculated. The results of these calculations are found below in the plot on the top of the next page. The maximum deflection is found at the tip of the beam and is found to be 2.91E-3 inches. This model assumes that the entire beam moves together and thus you only get a single value for displacement at each y location. Looking at the FEA results, shown below on the next page, we notice that this is not the case.

72
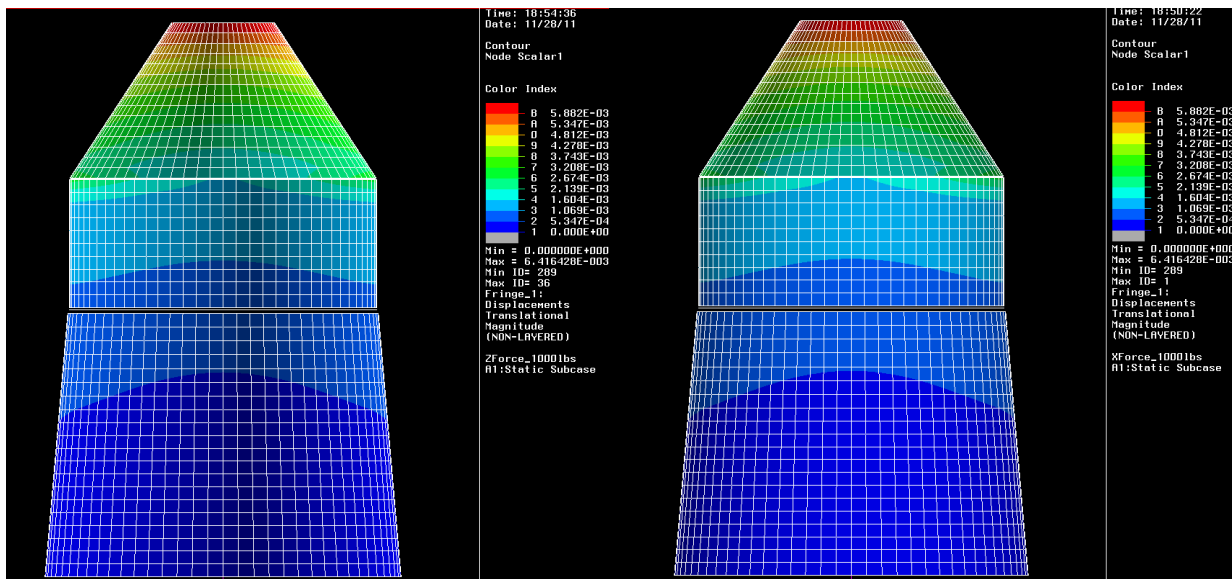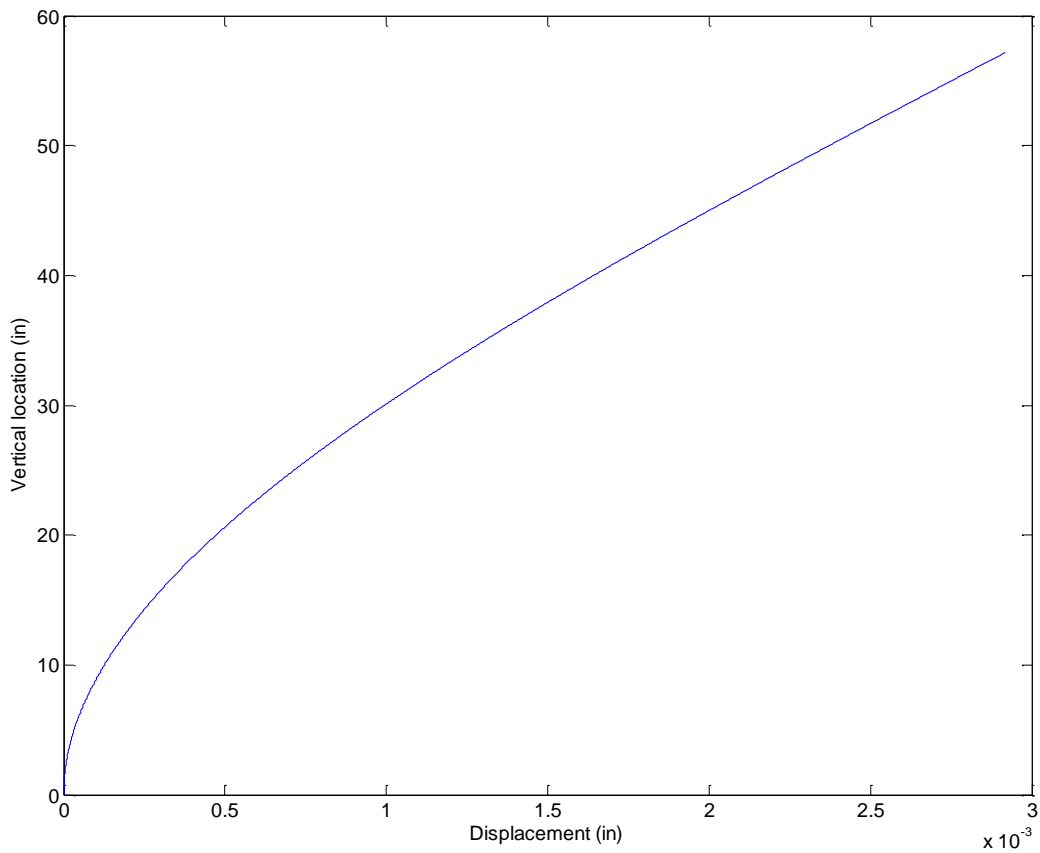
FEA results for 1000 lb axial load in the (a) X-direction and (b) Z-direction

The displacement is lower on the negative x/z (left side in the figure) side of the results. The higher fidelity causes the compression of the cylinder to be accounted for. The figure also shows that the maximum deflection is 6.42E-3 inches. This increase is likely due to that compression that is accounted for. There is movement in the x/z direction that is not accounted for in the simplified model. Even without that, the solutions are well within an order of magnitude difference from each other showing a strong correlation between the models.

Y-Moment Case

The third case we examined was 1000 in-lbs of pure torsion about the y axis. For this case we do not expect to see any bending displacement. The only displacement should be the twisting of the cylinder as a result of the force. To analyze the structure, we simplified the structure into three prismatic sections. Using the equation,

$$\overline{} \tag{4}$$

the twist angle was determined for the structure. A weighted average was used for the G and J values for each section. Using this equation yields a single solution at the point farthest from the constraints. It assumes that there is no internal distortion of the structure, i.e. the cylinder remains perfectly circular. Calculating the twist angle using this equation yields a solution of 2.14E-5 radians or .0012 degrees. From the twist angle we can solve for the displacement by calculating the arc length of the circle swept out by the twist angle using the equation:

$$\tag{5}$$

Since theta is very small we can use the small angle approximation to assume that the displacement is equal to the arc length, therefore:

$$\tag{6}$$

yields the final displacement to be 1.14E-4 inches at the forward ring.

When we look at the FEA results we find a displacement of 2.86E-4 inches at the forward ring. This gives a difference of approximately 60%. Considering the difference in assumptions, this shows that the two models do roughly match up. We did expect a difference since the simplified assumptions did not include the rings and simplified the values of G and J for each skin section as constant. In addition to the similarities, we find that the displacement looks constant for a given y location indicated by the concentric color bands on the structure shown in the figure below. This is congruent with our understanding of pure torsion cases; since the structure is symmetric, the stress and strain around the ring should have no concentrations. It should only vary with the geometric radius and material properties. The displacement should also be increasing with distance from the constraint location as it clearly does. With these indicators, this load case also suggests the appropriateness and correctness of our FEA model.
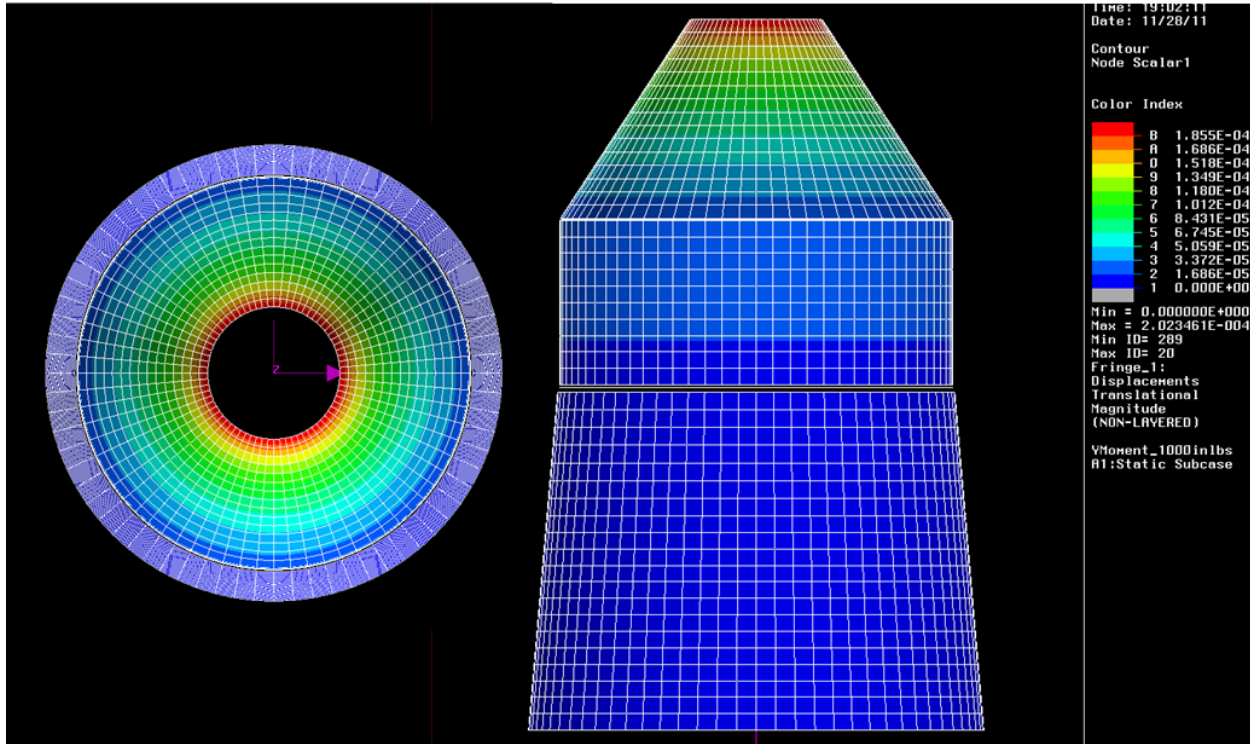
Figure 7. Displacement plot for y moment load case (Left: Top View, Right: Side View)

Z,X Axis Moment Case

The last cases we examined were the Z and X axis moment cases. As with the axial load cases, the z and x moment cases produce identical displacements that are rotated by 90 degrees. The stress distribution is found by using the stress equation:

$$\text{—} \tag{6}$$

To simplify the problem, this was only solved at the maximum moment arm, i.e. where x=r. This gives the solution for theta=0, furthermore, due to symmetry, the solution for theta=180 degrees, the solution is simply the negative of the value. The rest of the theta locations can be solved by using trigonometric functions to determine the moment arm for each point about the radius. It was not necessary to evaluate every point so this step was skipped. Once the stress was determined for each y location at theta, the local strain and displacements were found by using equation 2 and integrating over the length of the cylinder. In the analysis, we find that the maximum displacement is 2.97E-4 inches. By looking at the displacement as a function of the y location, shown in the plot located on the next page below, it is shown that most of the displacement, occurs in the upper skin section. This is shown by the lower slop in the upper part of the curve. This same trend is noted on the FEA results shown on the bottom of the following page. The color bands are much narrower in the upper skin indicating a rapid change in the displacement. In addition we note the maximum displacement. First of all is the magnitude at 3.93E-4, an increase of only 25% over our calculated levels. The proximity of the solutions indicates that the model is yielding an appropriate a solution that is in
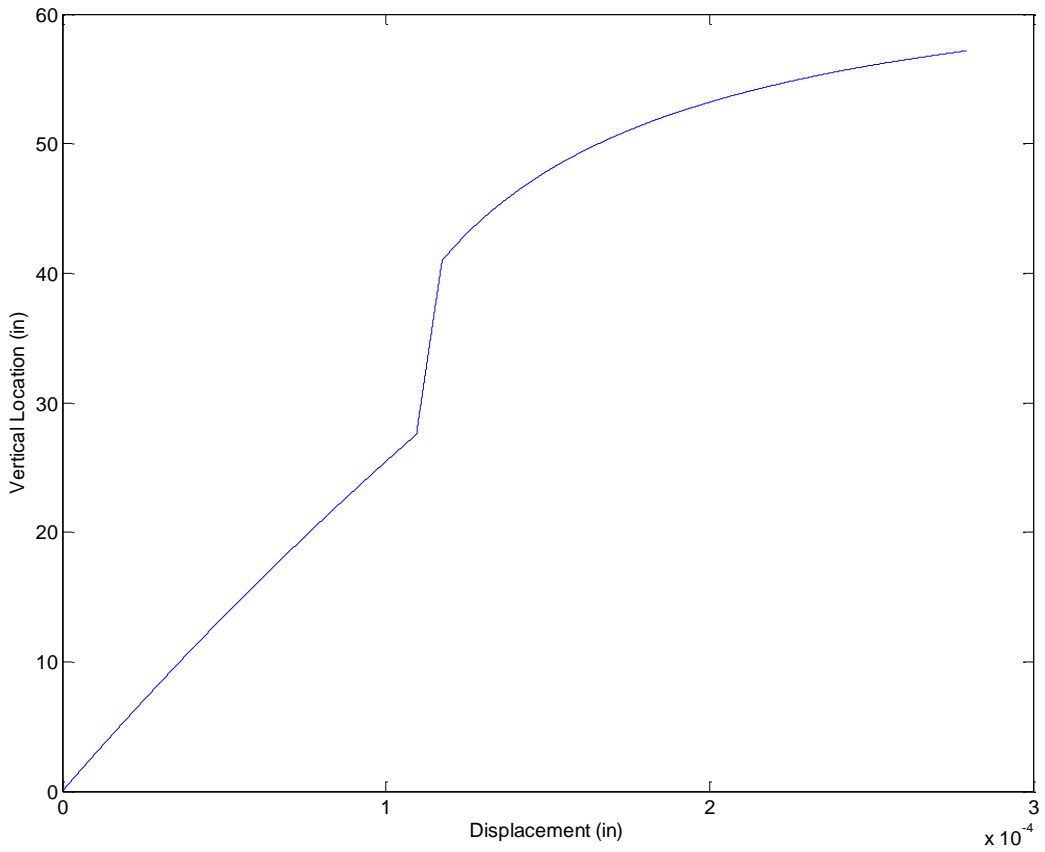
Calculated displacement for the Z,X axis moment cases at theta=0 location.
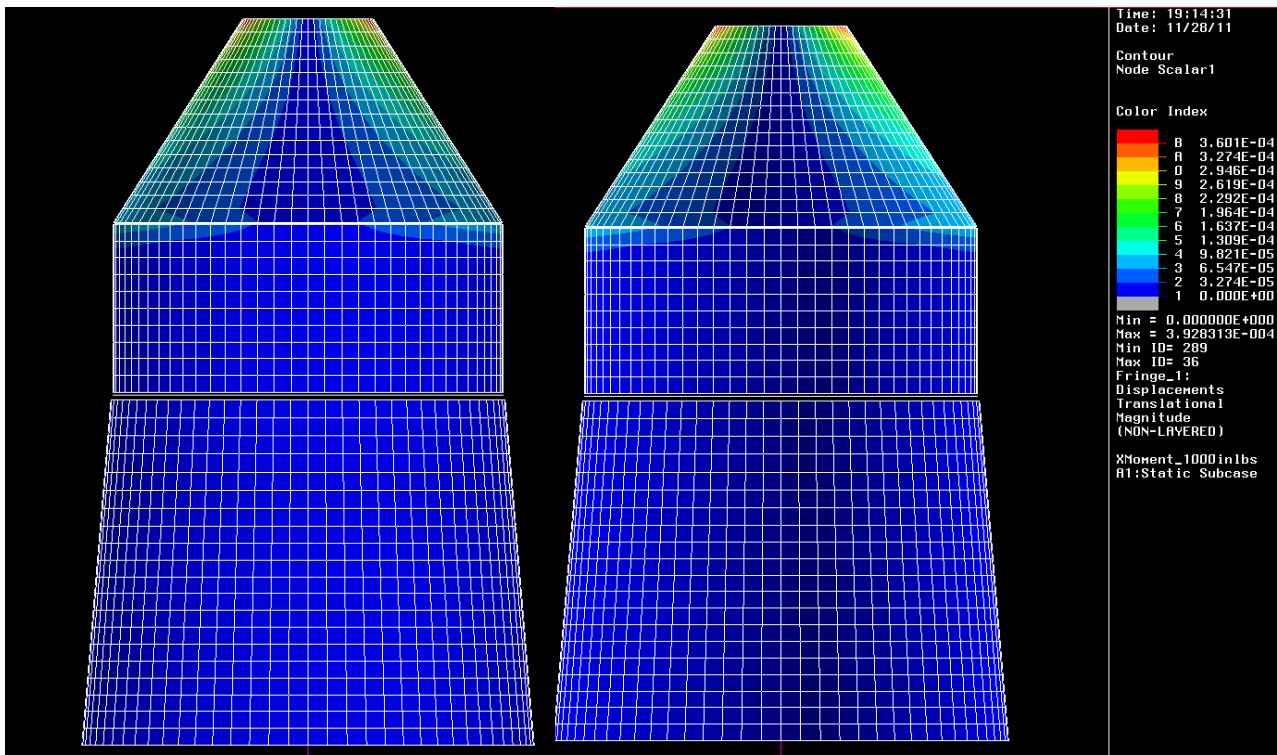
a)   b)

FEA results for (a) Z-axis (b) X-axis moment cases

congruence with our hand calculations.   More importantly, is the location of the maximum displacement. It is at theta=0 which is the exact same location as predicted by the hand calculations.

In addition to the values we calculated, we can look at the overall results and check to make sure they concur with our general understanding of structures. The theory and equations say that the stress should decrease with the normal distance from the point of application. Looking at the stress plot yielded by FEA, we note that the values given near the center are much lower than those at the extremities. In addition the displacement decreases as we approach the constraint location at the separation ring.   Both of these observations are in congruence with our understanding of structures helping to validate the model.

Analysis Results & Conclusion

While we cannot make a perfect FEA model, we can make certain simplifying assumptions that allow us to make a fairly accurate model. In order to validate our model, we must prove that it stands up to simple cases that we can calculate by hand. The summary of our results are shown in the table below. The case showing the most difference is the Y-moment case showing a 60% difference between the FEA and the hand calculation results, still less than an order of magnitude. By using these 6 test cases, each of which has shown a good correlation between FEA and hand calculation results, we have gained confidence that our model is both appropriate and correct to our assumptions. In order to increase our confidence with this model we could look at some more complicated or different load cases to ensure that the good correlations continue.

	Maximum Displacement		
Load Case	Hand Calculation (in)	FEA Results (in)	Percent Difference
Y-Axial	1.73e-3	1.95e-3	11%
X,Z Axial	2.91e-3	6.42e-3	55%
Y-Moment	1.14e-4	2.86e-4	60%
X,Z Moment	2.97e-4	3.93e-4	25%

References

Wertz, James Richard. "Structures." *Space Mission Analysis and Design*. El Segundo: Microcosm, 2003. Print.

Ugural, A. C. *Mechanics of Materials*. Hoboken, NJ: J. Wiley & Sons, 2008. Print.

Matlab Code

```
clc
clear all
close all
 y=0:.01:57.17;
deltaL=ones(size(y));
%_____
%% Lower Skin
ylow = 0:.01:27.53; %in
tlow = .38835; %in
rlow = 18.319-.085543.*ylow; %in
Alow = pi*(2*rlow*tlow+tlow^2); %in^2
Elow = 6.82e5; %psi
Glow = Elow/2.66; %psi
Ilow = pi*(rlow+tlow/2).^3.*tlow; %in^4
Jlow = 2*Ilow; %in^4
%_____
%% Central Skin
ymid = 27.54:.01:40.95; %in
tmid= .2; %in
rmid= 15.928*ones(size(ymid)); %in
Amid = pi*(2*rmid*tmid+tmid^2); %in^2
Emid = 10.4e6; %psi
Gmid = Emid/2.66; %psi
Imid = pi*(rmid+tmid/2).^3.*tmid; %in^4
Jmid = 2*Imid; %in^4
%_____
%% Upper Skin
y_up = 40.96:.01:57.17; %in
t_up = .035833; %in
r_up = 15.875-.6492*(y_up-40.96);
A_up = pi*(2*r_up*t_up+t_up^2); %in^2
E_up = 10.4e6; %psi
G_up = E_up/2.66; %psi
I_up = pi*(r_up+t_up/2).^3.*t_up; %in^4
J_up = 2*I_up; %in^4
%_____
%% Total Properties
A = [Alow,Amid,A_up];
r = [rlow,rmid,r_up];
E = [Elow*ones(size(Alow)),Emid*ones(size(Amid)),E_up*ones(size(A_up))];
I = [Ilow,Imid,I_up];
J = [Jlow,Jmid,J_up];
G = [Glow,Gmid,G_up];
```

```matlab
% plot(A,y)
% axis([0 60 0 60])
%_____
%% Y Axial Case

P = 1000; %lbs
sigma = P./A; %psi
epsilon = sigma./E;

deltaL(1) = .01*epsilon(1);
for i=2:length(epsilon)
    deltaL(i) = .01*epsilon(i)+deltaL(i-1);
end

% plot (deltaL,y)
% ylabel('Vertical Location (in)','fontsize',14)
% xlabel('Displacement (in)','fontsize',14)
 %_____
%% X,Z Axial Case
P = 1000; %lbs

deltax = -P.*y.^2/(6*mean(E)*mean(I)).*(y-3*max(y));

% plot(deltax,y)
% ylabel('Vertical location (in)','fontsize',14)
% xlabel('Displacement (in)','fontsize',14)

%% Y Moment Case
T = 1000; %in-lb

theta=T*(max(y_up)-max(ymid))/(mean(G_up)*mean(J_up))+T*(max(ymid)-
max(ylow))/(mean(Gmid)*mean(Jmid))+T*max(ylow)/(mean(Glow)*mean(Jlow))
%_____
%% X,Z Moment Case
M = 1000; %in-lb
sigma = M.*r./I; %psi
epsilon= sigma./E;

deltaL(1)=.01*epsilon(1);
for i=2:length(epsilon)
    deltaL(i)=.01*epsilon(i)+deltaL(i-1);
end

% plot (deltaL,y)
% ylabel('Vertical Location (in)','fontsize',14)
% xlabel('Displacement (in)','fontsize',14)
```

VI. Conclusion [Richard Pelham]

Requirements for Completion of the Model

The detailed Pro/E model of the Boeing 376 spun satellite is mostly complete. This model only has the thrust tube and spun shelf of the satellite as you can see from the figure below.
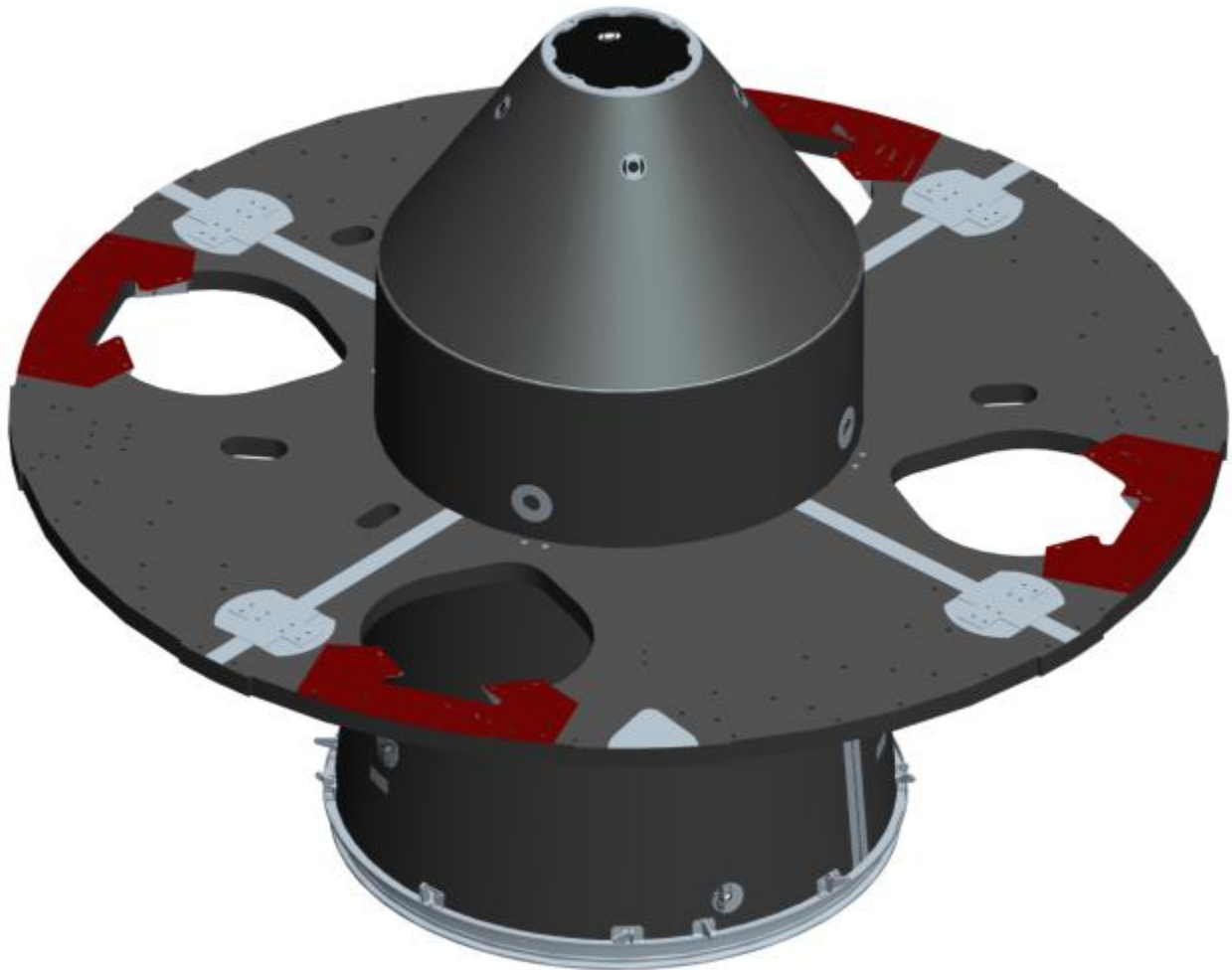
The model has a significant amount of detail. However, missing from the model are a number of doublers, fasteners, spacers, rivets, brackets and the beryllium supports between the spun shelf and the thrust tube. The engineering drawings for these should become available at a later date, and the beryllium supports will likely be a project for other students to perform in the future.

Also missing from the model is the de-spun section of the satellite. None of the drawings for this section were included in the package we received. Once these become available, they will need to be modeled and added to the spun section of the satellite.

In order to generate detailed drawings of this model at a later date, the dimensions and tolerances must be added to each of the parts and components. Most of the dimensions are already a part of the model; however, they are not immediately visible unless you access the sketches for each individual part. None of the tolerances were included since the model itself does not require them to visually reproduce the satellite. These tolerances are essential to creating a physical model of the satellite.

A detailed list of all the parts included in this model, and the parts that will need to be completed and added to this model at a later date, can be found on the Aerospace Engineering server under the space folder in the CPIntersep folder. This folder is only accessible with special permission.

Visual Issues

First, these are not problems with the model itself; it is just a concern for the visual aesthetics of the project. Part of this project required a detailed visual model of the Boeing 376 satellite. We made an effort to make the model look correct but we had some visual difficulties due to the way Pro/E renders the parts.

For an unknown reason, a few of the parts are not rendering correctly. This mainly deals with the doublers which are very thin sheets placed on the surfaces of the thrust tube. These doublers are placed correctly, but they appear as if they are entering the surface. More than likely this is because of the way Pro/E draws the models. As of this time, we cannot find a way to fix this. An example of this can be seen in the following figure.
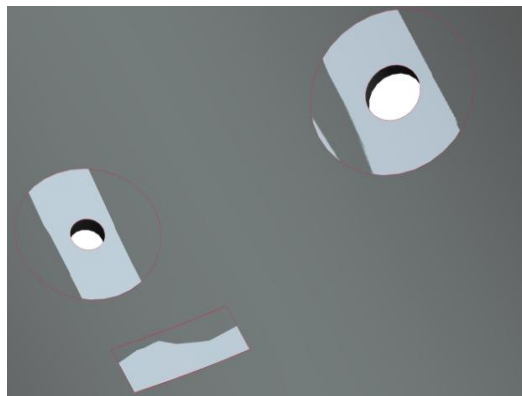
Another issue we are having with the rendering of the model can be seen in the figure below. The red outline shows the head of a screw, the washer and nut can also be seen. The shaft of the screw is missing though. This only occurs on a few times throughout the model. This may be an effect from using the pattern feature of Pro/E.
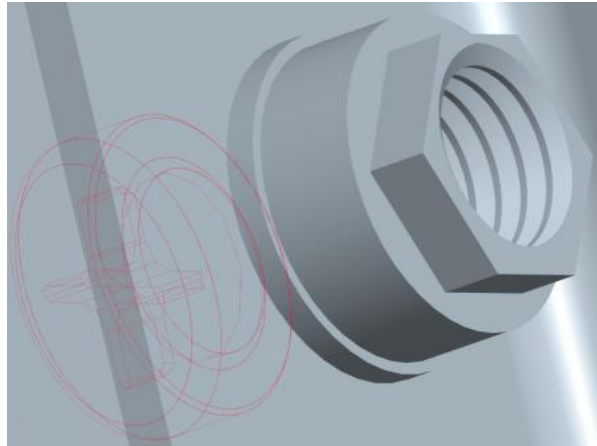
Possible Problems in the Future

The main axis of the pro/e model is not to industry standards. In order to correct this, the z and y axes must be interchanged. At this time, this is not an issue as all of the numbers used in both the detailed model and the FEA model take this into account. Once we start dealing with industry this may become a problem. There may be a way to change the axes without completely redoing the models; however, changing the axes may result in problems with both of the models. The numbers calculated and used in both PATRAN and Pro/E were found using the current axis set-up. Therefore, changing the axes may require a recalculation of the inertia matrix, and many of the offset values used.

Appendix

A. RELEVANT FILES AND LOCATIONS

	File Name	Description	Location
Parts List	Parts List.xlsx	File contains indentured parts list for spun and de-spun section including color coding for drawings received and parts modeled.	N:\Aero\CPinterSEP\Model\Parts List.xlsx
Drawings	Folder	Folder Containing all drawing files	N:\Aero\CPinterSEP\Model\Drawings
FEA Patran Database	TC6D_test.db	Patran Database file containing FEA model and results of test cases (this is the most up to date file) **NOTE: Must be opened from Patran 2008r2**	N:\Aero\CPinterSEP\Model\FEA\TC6\TC6D_test.db
Spun Structure Assembly(Pro/E)	7943088-001.asm	Pro/E Assembly file for spun section	N:\Aero\CPinterSEP\Model\Final Spun Assembly\7943088-001.asm
Spun Structure Assembly FEA (Pro/E file)	7943088-001_fea2.asm	Pro/E assembly file reduced for export into Patran for FEA	N:\Aero\CPinterSEP\Model\Final Spun Assembly\7943088-001_fea2.asm
Pro/E Parts	Folder	Folder containing all of the latest part and assembly files	N:\Aero\CPinterSEP\Model\Final Spun Assembly

B. ABET CRITERIA PREFACE [Kevin Povey]

The Accreditation Board for Engineering and Technology ensures the quality of postsecondary education for students. ABET was established in 1932 and includes 28 professional and technical Societies including AIAA- The American Institute of Aeronautics and Astronautics. Among other minimum curricula specified by ABET, a capstone project or design class is required for each student. This project must meet a number of criteria in order to ensure graduates have had experience and a minimal skill set for entry into the workforce or future education. During the preliminary planning and research phase of our project, a number of criteria have been completed with some still in progress and others not yet completed.

At this point in the project we have already *shown our ability to function on a multidisciplinary team.* Each member of the group is an aerospace engineering student, yet we have had to interface with members of different professional fields. It has involved working with an Industrial-Manufacturing Engineer (IME) to learn the 3-d modeling capabilities and practices for Pro/Engineer of an Industrial-Manufacturing Engineer (IME). In addition, each group member has different strengths and background, from which they have had to share and collaborate with other members. Functioning on a multidisciplinary group will continue to be important as the project goes on in order to get IME expertise on properly interpreting part and assembly blueprints along with ME knowledge on the process of conversion into finite element analysis models. In addition we have already completed the criteria of *identifying, formulating and solving engineering problems.* Members of the group have contacted appropriate industry professionals and local faculty in order to provide guidance for proper methods for rendering 3D solid models and assembly as well as for the integration for any parts from the Pro/Engineer software to PATRAN.

There are also multiple ABET criteria that the group is currently in the progress of satisfying. The most significant is *designing a system, component, or process to meet desired needs.* We have been working on developing an organization structure or PDM in order to regulate and distribute the different parts and assemblies of the total satellite. The system is being created using Microsoft Office access and will function similar to a library checkout system. In addition, we are in the process of working on *effective communication.* So far, communication has consisted of planning within the team, contact and discussions with the project advisor, and interactions with those leading the CPInterSEP program. At this point we have not had any direct communication with the Boeing Company and instead have been getting the information through our project advisor. Effective communication will continue to play a very important role in the project, both in managing part assignments within the team, and in staying up to date with CPInterSEP. We are also in the progress of satisfying the criteria of *using modern engineering tools, skills and techniques.* A large portion of this past quarter has involved getting familiar with the Pro/Engineer modeling software and the Patran finite element analysis conversion software.

These are both very effective engineering tools and will play a primary role in the completion of this project. We are, however, still in the process of learning the most effective ways of using these programs.

Finally, there are a number of the ABET criteria that though they have not yet been satisfied, we hope to complete before the conclusion of the project. First, we are going to *apply our knowledge of mathematics, science, and engineering*. In order to correctly model the parts, it will be necessary to have an understanding of engineering design documents. We will have to correctly interpret the drawings and transfer the features into a 3D model. In addition, we will need to perform some basic analytical stress analysis on the structure based on projected loads in order to have a comparison for the FEA models. Second, we plan to *design and conduct experiments, as well as analyze and interpret data.* If time allows, we are going to import the satellite assembly into PATRAN and perform an FEA analysis. The resulting data will show the stresses and how the completed assembly will handle the test load environments. Interpreting the data will be critical in ensuring that the solid model has been appropriately generated and can be correctly imported into the FEA software. Third, we will show that we *understand our professional and ethical responsibilities.* We are going to be receiving proprietary blueprints and drawings of the structure and assembly of the Boeing 376 spacecraft which are subject to the controls of the International Traffic in Arms Regulation (ITAR). Thus, the blueprints of these parts must be handled in a professional manner so as to control the release of these documents to authorized individuals or organizations. Also, properly producing a completed assembly model and finite element analysis in an honest manner is necessary to mitigate incongruities between test results for the model and physical spacecraft structure in assembly efforts for the future. Fourth, we hope to *show the impact of engineering solution in a global and societal context.* The project involves setting a model base for future members working on the 376 project. It will demonstrate the proper processes necessary for adding parts to the assembly and creating a model that is compatible with PATRAN and finite element software. Lastly, our project will *demonstrate the life-long learning process.* The work done on this project will have an impact on the possibility of future experimental instrumentation in space. The modeling of the primary structure of this spacecraft will be used to analyze the stress implications of adding various experiments. This project will set the base for the reality of eventually launching the satellite.

C. ABET CRITERIA COMPLIANCE [Kelly Cheng & Kevin Povey]

✓ *Apply knowledge of mathematics, science, and engineering*

An understanding of engineering design documents will be necessary to correctly model parts in the solid modeling software. The material properties pertaining to all satellite parts will be researched for proper structural characteristics of the completed assembly. Analytical stress analysis will be performed based on projected loads. Numerical stress analysis will be executed using finite-element analysis (FEA) tools to validate and support analytical results.

✓ *Design and conduct experiments, as well as analyze and interpret data*

The satellite structure assembly will be imported into PATRAN and FEA results will display computerized stress calculations. The data will reveal how the completed assembly will handle the test load environments. Interpreting the data will be critical in ensuring that the solid model has been appropriately generated and can be correctly imported into the FEA software.

✓ *Design a system, component, or process to meet desired needs*

A proper database structure will need to be designed and implemented to organize the various part files necessary for the completed satellite assembly. Create a baseline structure that will function correctly and have the ability to be modified to include future changes to satellite structure. The proper generation of the satellite assembly model will allow for the streamlining of a process to import the assembly into the FEA analysis tool.

✓ *Show ability to function on multi-disciplinary team*

This endeavor will include a team of several engineers operating under multiple disciplines. The project will involve the 3-d modeling capabilities of an Industrial-Manufacturing Engineer (IME). IME expertise will also be necessary to properly interpret part and assembly blueprints. The abilities of an Aerospace/Mechanical Engineer (AERO/ME) will also be necessary in order to demonstrate the process of conversion into a finite element analysis model is functioning correctly.

✓ *Identify, formulate and solve engineering problems*

Contact appropriate industry professionals and local faculty in order to provide guidance for proper method to render 3-d solid models and assembly as well as for the integration of a spacecraft assembly from the Pro/Engineer software to PATRAN. If blueprints for any parts to the satellite primary structure are unavailable, measurements of the respective parts, available on-site, will need to be taken using measurement tools.

✓ *Understand your professional and ethical responsibilities*

The proprietary blueprints available for reference for the compliation of this project are subject to the controls of the International Traffic in Arms Regulation (ITAR). Thus, the blueprints of these parts must be handled in a professional manner so as to control the release of these documents to authorized individuals or organizations. The reduction of waste and cost is a responsibility belonging to each member of this team. Properly producing a completed assembly model and finite element analysis in an honest manner is necessary to mitigate incongruities between test results for the model and physical spacecraft structure in assembly efforts for the future.

✓ *Show effective communication*

When working on a team consisting of five members, effective communication between each person will be essential. Each task will require information that will need to be provided another member, and it is important to have a constant flow of information. With a vast number of individual tasks, it will be important to keep track of who is doing what. In addition, it will be necessary to be in contact with industry personnel. We will need to communicate our questions and needed information with the Boeing Company in a professional and effective manner.

✓ *Show impact of engineering solution in a global and societal context*

The project involves setting a model base for future members working on the 376 project. It will demonstrate the proper processes necessary for adding parts to the assembly and creating a model that is compatible with Patran and finite element software.

✓ *Demonstrate life-long learning process*

The work done on this project will have an impact on the possibility of future experimental instrumentation in space. The modeling of the primary structure of this spacecraft will be used to analyze the stress implications of adding various experiments. This project will set the base for the reality of eventually launching the satellite.

✓ *Use modern engineering tools, skills and techniques*

This project relies heavily on modern engineering tools. The main bulk of the project will be done on the ProEngineer modeling software. Analytical stress analysis calculations will be performed and compared to results gained from finite element analysis software. In addition, Patran be used to make finite element analysis possible with the ProE Models.

D. PEER EVALUATION CRITERIA & FORM [Jeffrey Ma]

To evaluate other group member's performances we will be grading each other on seven different areas. The seven different areas are meeting attendance, meeting participation, work ethics, work timeliness, work contribution, work quality and effective communication. To grade each other, in these areas, we will be using a number scale from one to five, one being 'Great' and five being 'Greatly Needs Improvement.' In addition to these seven areas we will also suggest a grade which each individual deserves and comment as to why they deserve these grades.

Name	Attendance	Participation	Ethics	Timeliness	Contribution	Quality	Communication	Grade	Comments/Reasons
Jason Carpenter									
Kelley Cheng									
Jeffrey Ma									
Kevin Povey									
Richard Pelham									
*To rate people please use this scale: 1 ~ Great, 2 ~ Good, 3 ~ Average, 4 ~ Needs Minor Improvement, 5 ~ Greatly Needs Improvement									

E. MANAGEMENT STRUCTURE & CONFIGURATION CONTROL
[Jason Carpenter]

The management structure will be broken down by task as seen in the management structure outlined below in fig. 2. Jason Carpenter will be serving as project manager and configuration control. We will have everybody working on the modeling and assembly. The bulk of the modeling will be done by our modeling team which will consist of Kelly Cheng, Jeffrey Ma, and Kevin Povey. Their exact tasks will be distributed once we receive the indentured drawing list and detail drawings. Richard Pelham will lead up the assembly effort with assistance from Jason Carpenter and the modeling team as needed. In addition to assemblies, Richard will
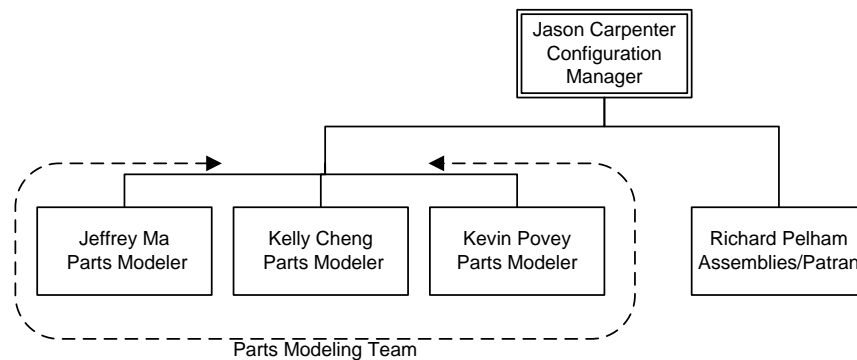
also be in charge of checking the model for Patran compatibility.

In order to maintain control of the model and make sure that the parts are not redundantly created, we will be imposing a configuration control system. Without access to a Product Data Management (PDM) system we will be creating our own version using Microsoft Access and a configuration manager. Jason will be serving as configuration manager. He will control the official model and check out parts to a single user at a time. The official copy of the model will be maintained by Jason on his server partition. To maintain quick access to the model at all times, a public copy of the model will be maintained on the CPInterSEP server. The purpose of this copy is to allow access to the model at all times. Even though all team members have access to the parts, they will not be allowed to make official changes without first checking out the part though the configuration manager. Parts will not become official however until returned to the configuration manger and accepted as official changes. Once a change is accepted, the revised model will be published to the server.

An Access database will be used to track changes and details for each of the parts. We will be using a check-in/check-out system to ensure that parts are not being edited by multiple users at one time. We will also be tracking many details of the parts description and history including the following:

- Part ID (unique identifier given by Access)
- Boeing Part Number (including revision)

- Part Description – A brief physical description of the part
- Person Checked out To
- Check Out Date
- Expected Return Date
- Latest Revision Date
- Model Revision History – Revisions to the drawings
- Subordinate Parts/Assemblies (for Assemblies)

F. GANTT CHARTS [Jason Carpenter]

Winter Version

	ⓘ	Task Name	Duration	Start	Finish
1	✓	⊟ **Project Conception/Setup**	**1 day**	**Mon 1/25/10**	**Mon 1/25/10**
2	✓	Gantt Chart	1 day	Mon 1/25/10	Mon 1/25/10
3	✓	Determine Group Grading	1 day	Mon 1/25/10	Mon 1/25/10
4	✓	Determine Individual responsibilities	1 day	Mon 1/25/10	Mon 1/25/10
5	✓	⊟ **Research**	**20 days**	**Mon 1/25/10**	**Fri 2/19/10**
6	✓	Patran	20 days	Mon 1/25/10	Fri 2/19/10
7	✓	Nastran	20 days	Mon 1/25/10	Fri 2/19/10
8	✓	.step/.iges files	20 days	Mon 1/25/10	Fri 2/19/10
9	✓	accuracy	20 days	Mon 1/25/10	Fri 2/19/10
10	✓	Material Properties	20 days	Mon 1/25/10	Fri 2/19/10
11	✓	Get Secure Server Partition	14 days	Tue 1/26/10	Fri 2/12/10
12	▦	Determine and Setup Organizational Model	14 days	Mon 2/22/10	Thu 3/11/10
13	▦	Write Preliminary Report	6 days	Fri 3/12/10	Fri 3/19/10
14		Determine Modeling Priorities/Order	14 days	Mon 3/22/10	Thu 4/8/10
15	▦	Get Drawings	1 day	Fri 4/9/10	Fri 4/9/10
16	▦	Model Parts	30 days	Mon 4/12/10	Fri 5/21/10
17	▦	Assemble Parts	30 days	Mon 4/12/10	Fri 5/21/10
18	▦	Setup for FEA model transfer	33 days	Mon 4/19/10	Wed 6/2/10
19		Complete Model	1 day	Thu 6/3/10	Thu 6/3/10
20	▦	Write Project Report	6 days	Fri 6/4/10	Fri 6/11/10
21	▦	Project Report Draft Completed	1 day	Fri 6/11/10	Fri 6/11/10
22	▦	Revise Report	2 days	Fri 6/11/10	Sun 6/13/10
23	▦	Project Report Final	1 day	Mon 6/14/10	Mon 6/14/10