2019-01-01

Thermal/Mechanical Analysis and Development of a 2000 lbf Liquid Oxygen/Liquid Methane Rocket Engine

Marissa Garcia
University of Texas at El Paso

THERMAL / MECHANICAL ANALYSIS AND DEVELOPMENT OF A

2000 LBF LIQUID OXYGEN / LIQUID METHANE

ROCKET ENGINE

MARISSA BEATRICE GARCIA

Master's Program in Mechanical Engineering

APPROVED:

_____

Jack F. Chessa, Ph.D., Chair

_____

Ahsan Choudhuri, Ph.D.

_____

Luis Rene Contreras-Sapien, Ph.D.

.

_____
Stephen Crites, Ph.D.
Dean of the Graduate School

Dedication

I dedicate this thesis to my God, my family, and to my friends. My God for giving me the
strength when I was at my weakest, my parents for providing unconditional love and support and
a hug when I needed it most, and to my friends for motivating and believing in me throughout
the years. You know who you are, and I thank you. ♪

THERMAL / MECHANICAL ANALYSIS AND DEVELOPMENT OF A

2000 LBF LIQUID OXYGEN / LIQUID METHANE

ROCKET ENGINE

by

MARISSA BEATRICE GARCIA, B.S.M.E

THESIS

Presented to the Faculty of the Graduate School of

The University of Texas at El Paso

in Partial Fulfillment

of the Requirements

for the Degree of

MASTER OF SCIENCE

DEPARTMENT OF MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT EL PASO

December 2019

Acknowledgements

First off, I would like to give a huge acknowledgement to Dr. Ahsan Choudhuri and Dr. Jack Chessa for their continued guidance and support throughout my research career at the Center for Space Exploration and Technology Research (cSETR). They both have challenged me to get out of my comfort zone and have provided opportunities that helped me grow as an engineer. The experience that I gained from working at cSETR is unparalleled and is something I hope many future students have the opportunity to experience as well.

Secondly, I would also like to acknowledge and extend my sincere gratitude to Luz Bugarin, Gloria Salas, Scott Hill, Dr. Joel Quintana, and Brenda Sanchez. Mr. Hill, you have been one of my greatest mentors and I thank you for your candor, for the advice, and for always believing in me. Thank you, ladies, for your guidance, patience, conversations, and the laughs.

Lastly, I would like to acknowledge my amazing team members and friends: Mariana Chaidez, Linda Hernandez, Rene Miranda, Adrian Welsh, Gerry Ramirez, Alai Lopez, Manny Herrera, Steven Torres, Corey Hansen, Justin Vanhoose, Ray Rojo, Debbie Ortega and the rest of my colleagues that I have worked with and learned from in the past four years. This journey has been far from easy, but the hard work and long hours we have dedicated to our research is finally coming to fruition. We're finally seeing all the CAD and analysis models come to life! Thank you all for making my cSETR experience one full of laughs, tears, and everything in between. For this, I am eternally grateful.

Abstract

The Center for Space Exploration and Technology Research (cSETR) is developing three rocket engines: a 5 lbf reaction control engine, a 500 lbf engine, and a 2000 lbf engine. All these engines utilize a liquid oxygen and liquid methane propellant combination. The reason methane was selected for the fuel is because methane is easy to store, is self-pressurizing, and readily available. There is also the possibility of creating methane on planets such as Mars through an electrolysis process on the ice water found on Mars.

The 2000 lbf engine, or the CROME-X, is a modular, steady-state pintle injector engine. It is also throttleable on a 4:1 ratio. The design is currently undergoing both mechanical and thermal analysis. The analysis is being conducted both in Altair Hypermesh software and in Abaqus CAE. In Altair Hypermesh, the analysis being conducted by component; a thermal and mechanical analysis for the combustion chamber and a thermal and mechanical analysis for the injector. In the Abaqus CAE analysis, the entire thrust chamber assembly is modeled and uses contact surface interaction.

This thesis describes how to create the CROME-X models in both Hypermesh and Abaqus. It goes into detail on the boundary conditions used (symmetry and displacement constraints), the mechanical loads used (pressures and preloads), and the thermal loads (surface temperature and convection interaction) and how to model them in the two softwares in a step-by-step method.

Table of Contents

List of Tables

List of Figures

Chapter 1: Introduction

1.1 PURPOSE

The Center for Space Exploration and Technology Research (cSETR) has been developing liquid oxygen and liquid methane propulsion systems. This propellant combination was selected because of the possibility of in-situ resource utilization (ISRU). Robotic missions to Mars have recently reported that there is ice in subsurface deposits. This ice can be mined, and hydrogen and oxygen can be extracted from the ice to produce oxygen and methane to use as propellants, drinkable water, and oxygen for life support systems. Several aerospace companies such as Blue Origin and SpaceX are also investigating this propellant combination. Blue Origin is developing their BE-4 engine and SpaceX is developing their Raptor engine to demonstrate the usage of these propellants. In hopes of demonstrating this same technology, cSETR is currently testing both a 5lbf thruster and a 500 lbf engine, respectively known as reaction control engine (RCE) and the Centennial Restartable Oxygen Methane Engine (CROME), at the tRIAc facility in Fabens, TX. Along with this test campaign, cSETR is also in the developmental phase of a 2000 lbf engine known as the Centennial Restartable Oxygen Methane Engine eXtreme (CROME-X). The CROME will serve as the main engine for the Daedalus suborbital demonstrator, which is another project at cSETR. Daedalus will be launched out of Wallops Flight Facility and will coast past the Karmen Line, also known as the edge of space. Once past the Karmen Line, the vehicle will run maneuvers with six of the 5 lbf thrusters which are used for attitude control. If this vehicle successfully runs the maneuvers in space, Daedalus will become the first vehicle to use a liquid oxygen and liquid methane engine. The CROME-X will be the main engine for the Janus landing vehicle and this design could be potentially used for a lunar lander design. The mission for the Janus vehicle is to hover several feet in the air for a

desired amount of time, conduct roll maneuvers using four RCEs, and then touch back down for landing.

1.2 SCOPE OF THESIS

This thesis will go into detail about the details of the CROME-X engine characteristics, how to create a finite element analysis (FEA) model for a simplified generic engine model, and the details of developing a finite element analysis (FEA) model for the CROME-X.

The details of the CROME-X include information on the engine overview, as well as the designs for the injector and combustion chamber. The engine overview details the mission of CROME-X and the engine assembly, showing all the components that are a part of the assembly. The injector design discusses a brief overview, the bolt pattern and preloads needed to seal, and the redesign of the acoustic cavities. The combustion chamber section provides an overview of the chamber design.

The simplified engine model is primarily used to expediate the trouble shooting process due to the simplified geometry compared to the geometry of the CROME-X. The trouble shooting process of an FEA model requires a lot of time to reach a running process. The process on creating this simplified engine model includes a detailed procedure on how to use Hypermesh to divide components into multiple solids, create component hexahedron meshes, and create contact surfaces. Altair Hypermesh and Abaqus CAE were the analysis software used to create these models. Although Altair Hypermesh has the capability to model mechanical and thermal analysis, it was decided to use Abaqus CAE to model these analyses because of its greater capabilities. Altair Hypermesh will be used to create model symmetry, the mesh, and the contact surfaces. This quarter symmetry model that includes the mesh, assigned material properties, assigned property cards, and contact will then be inputted into Abaqus CAE where all mechanical and thermal loads will be applied.

The final sections provide guidance on how to construct the actual FEA model for the CROME-X in both Hypermesh and Abaqus. In both sections, there is detail on the boundary

conditions that are applied to the model. The boundary conditions include symmetry constraints, displacement constraints, preloads, pressures, surface temperatures, and convection interactions. These explanation behind the boundary conditions is also included.

Chapter 2: Engine Design

2.1 ENGINE OVERVIEW

2.1.1 Mission

The CROME-X is 2000 lbf throttlable, steady-state engine and has a pintle injector design. This engine will implement a LOX/LCH4 propulsion system and has a throttle ratio of 4:1, meaning it will throttle from 500 lbf to 2000 lbf. This engine is also designed to operate at mixture ratio (MR) of 2.7. The reason 2.7 was selected as the design MR is because this MR produces tanks of equal size for the propellants. This happens because at this MR, the liquid oxygen is approximately 2.7 times greater than that of the liquid methane density. Having equal tank sizes reduces the complexity of the vehicle because the tank designs will be the same.

2.1.2 Engine Assembly

The CROME-X engine assembly includes three main components, the combustion chamber, the injector body, and the pintle manifold. The engine assembly is a modular design to allow for opening and inspecting the engine during testing if issues arise. The modularity also allows for changing out pintle designs for example. The later CROME-X designs will be a single additively manufactured injector component and a thrust chamber. The test engine assembly can be seen in Figure 2.1 and the exploded view of the thrust chamber assembly can be seen in Figure 2.2.
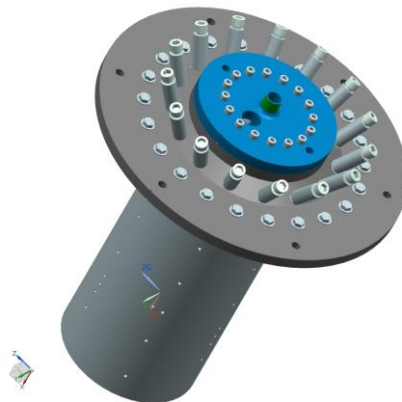
Figure 2.1 - CROME-X Engine Assembly

4
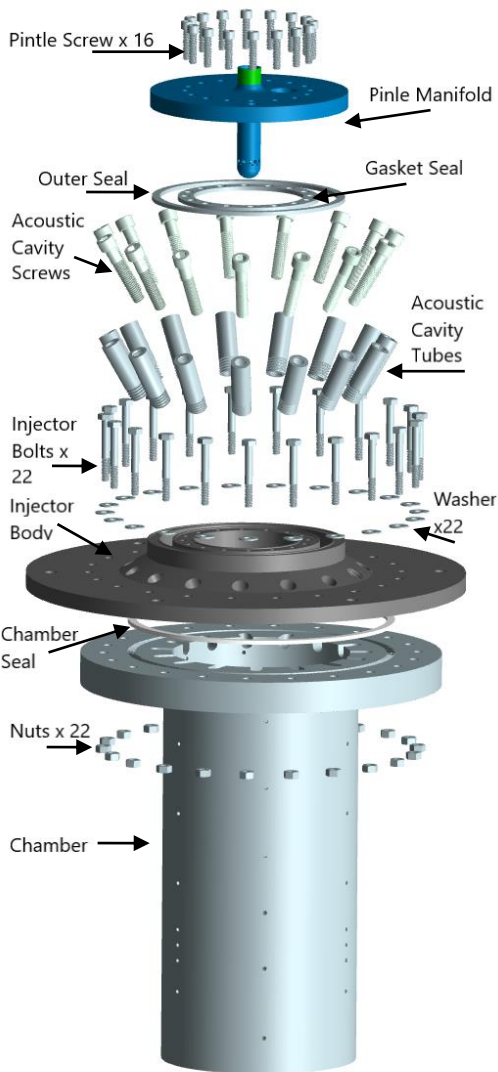
Figure 2.2 - Exploded view of the CROME-X assembly

2.2 INJECTOR DESIGN

2.2.1 Overview

The CROME-X engine utilizes an Inconel 625 pintle injector design and has acoustic cavities to dampen combustion instabilities. The pintle design was selected because the pintle injectors have a history of combustion stability and possess throttle capability [10]. Since CROME-X is a developmental engine, maintaining combustion stability is important for the safety of the students operating the tests and the safety of the hardware itself. The pintle post has a total of 24 holes which were sized based on the desired liquid oxygen injection area. The

injector is liquid oxygen centered, meaning that the liquid oxygen flows the pintle and enters the combustion chamber radially through the 24-hole injection area. The pintle has two sets of orifices for the injection area, the primary and the secondary. The primary orifices should inject most of the liquid oxygen from the post and the secondary injects the oxygen that the primary missed. These orifices can be seen in Figure 2.3. The liquid methane enters the combustion chamber axially via the annulus.
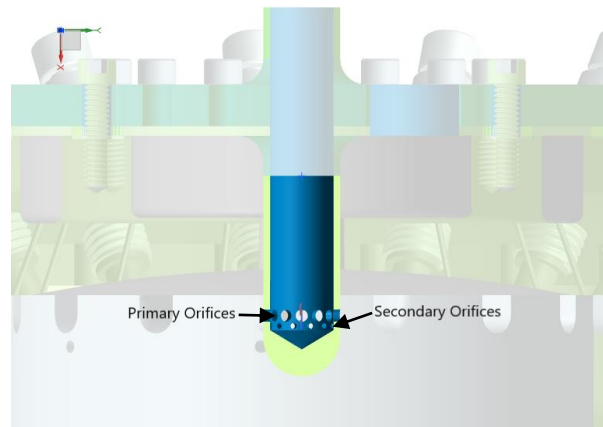
Figure 2.3 - CROME-X pintle post orifices

The methane flows along the outer part of the pintle post until it intersects with the liquid oxygen in the chamber. The fuel film cooling (FFC) is in a separate manifold and enters the chamber via a channel. The purpose of the FFC is to flow along the chamber walls and cool the chamber since it will be seeing combustion temperatures. The FFC will also contribute the combustion process as it atomizes. The flow of the propellants through the injector can been seen in Figure 2.4. The red arrows indicate methane flow and the green arrows represents oxygen flow. The scope of the design of CROME-X can be found in "*Design of a 2000 lbf LOX/LCH4 Throttleable Rocket Engine for a Vertical Lander* I. Lopez, 2017." [8]
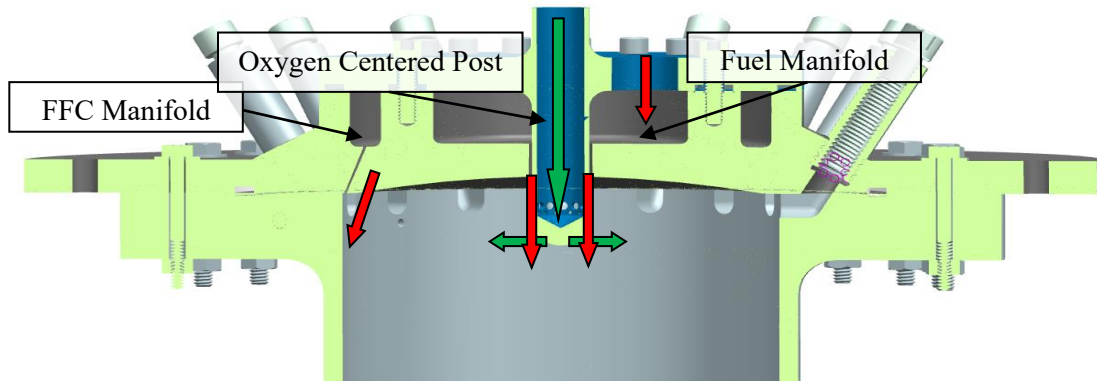
Figure 2.4 - CROME-X propellant flow path in the injector

The table below lists the desired injection areas for the oxygen, methane, and FFC that are needed to achieve the goal mixture ratio (MR). The table also lists the respective diameters for the pintle post orifices, and the pressure drops across the injector. These dimensions should not be changed during testing. The way to check if the injection area has changed during testing it to constantly monitor the inlet manifold pressures for oxygen, methane, and the FCC. If there is a significant drop in pressure ($\sim$ >15 psi), then the hardware may have been damaged, indicating the injection area dimensions increased. If the pressure increases, this could mean there's a clog in the system which needs to be cleaned.

Table 2.1 - Injection areas and pressure drop

Parameter	Value		
Injection Areas	LOX: 0.165 in^2	Fuel: 0.100 in^2	FCC: 0.043 in^2
Delta P Across Injector (2000 – 500 lbf)	70 – 7 psi		
dP/Pc (2000 – 500 lbf)	30.0 – 9.9%		
# of FCC orifices / diameter	15 / 0.061in^2		
# of Pintle Orifices / Orifice Diameters	Primary: 12 / 0.118 in^2		Secondary: 12 / 0.06 in^2

2.2.2 Bolts/Screws

There are two sets of bolts/screws in the engine assembly, the injector body to chamber bolt connection and the pintle to the injector body screw connection. Between each of these connections is a GORE seal that requires a certain amount of pressure to ensure that it does seal. GORE specifies that 2500 psi is the minimum pressure that must be applied to the Teflon for it to

seal. The table below lists the bolt and screw size, the number of bolts and screws, preloads, and the bolt torques. These calculations were made following the bolt preload load section in Shigley's Mechanical Engineering Design. [2]

Table 2.2 - Bolt and screw connections and preloads for sealing

Interface	Bolt Size	Number of Bolts	Preload (lbf)	Torque (in-lb)
Pintle Manifold →Injector Body	$\frac{1}{4}" - 20$	16	3377	152
Injector Body → Chamber	$\frac{1}{4}" - 20$	22	2115	95

The following figures are the bolt patterns on each component and show the order in which to tighten each bolt or screw. Using the torque wrench, torque each bolt 10 in-lb in the designated order until all the bolts or screws have achieved their required torque.
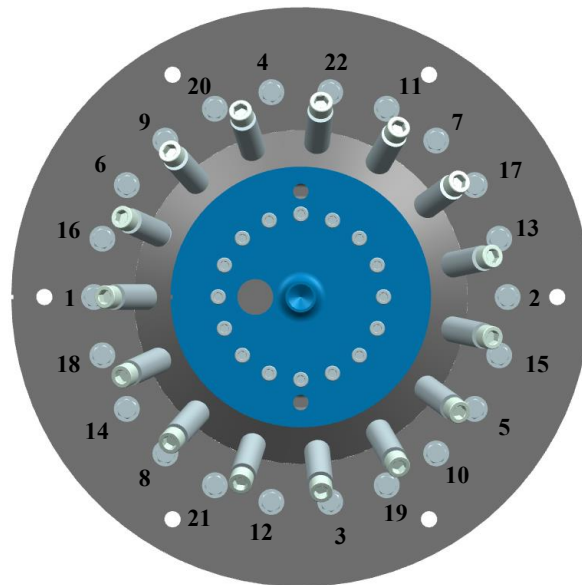
Figure 2.5 - Bolt preload order from the injector bolts
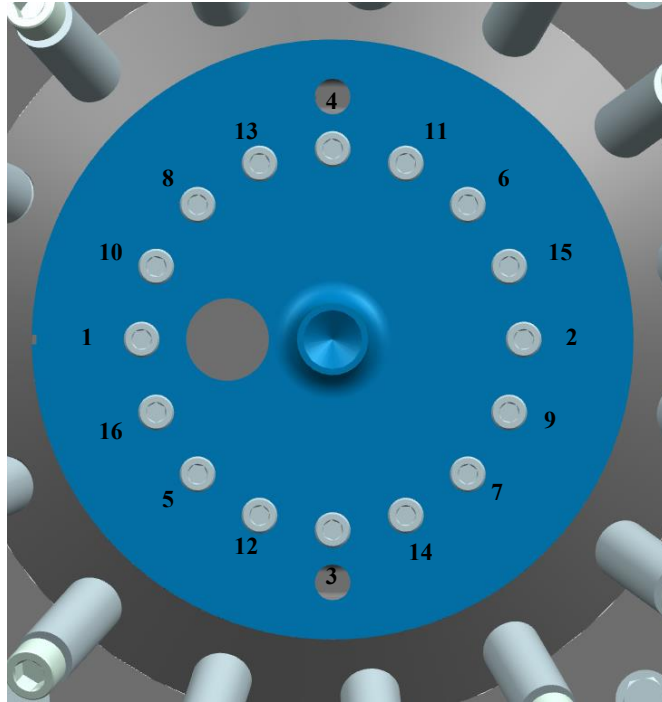
Figure 2.6 - Bolt preload order for the pintle manifold screws

2.2.3 Acoustic Cavities

Combustion instability is a frequent problem seen in liquid propellant rocket engines. These instabilities can decrease engine performance, create uncontrolled impulse, and can vary the direction of thrust (thrust vector). The types of instabilities that can occur are low frequency, intermediate frequency, and high frequency. The low frequency (<700 Hz), or "chug", occurs when oscillations in the feed system develop. These instabilities can be mitigated by decreasing the chamber volume, increasing the ratio between length and diameter in the supply/feed system, and by increasing the pressure drop across the injector. The intermediate frequencies (200-1000 Hz), or "buzz", occur when there is a coupling between the combustion process and the supply system. These frequencies could affect the mixture ratio which could then negatively affect engine performance. The high frequencies, or "screech", are caused by the engine itself rather than by the feed system. If these higher frequencies are reached because the oscillations were not dissipated, this could lead to hardware and instrumentation failure, or possibly engine explosion [5]. These instabilities result from the "coupling of the combustion process and the

9

fluid dynamics of the engine system." [9] These high frequency combustion instabilities are tangential (1T, 2T, 3T), radial (1R, 2R), or a combination of the two modes (1T1R, 1T2R). To prevent these acoustic instabilities, acoustic absorbers or baffles are implemented into the design to reduce the amplitudes of damaging frequencies. Acoustic absorbers are most commonly used to remedy combustion instabilities. [3]

The acoustic cavities for the CROME-X were redesigned because the original design had one continuous, circular volume within the injector body and used acoustic blocks for dampening. This open volume injector coupled with a large temperature gradient caused by combustion and cryogenic propellant temperatures created stresses in the injector that surpassed the required safety factor. One method of reducing stress is to thicken up the component. The new acoustic cavity design implements this idea by increasing the amount of material in the injector component. The new design consists of 15 x 3/8" diameter acoustic cavities as seen in Figure 2.7. A single acoustic cavity is boxed in black in the figure below.
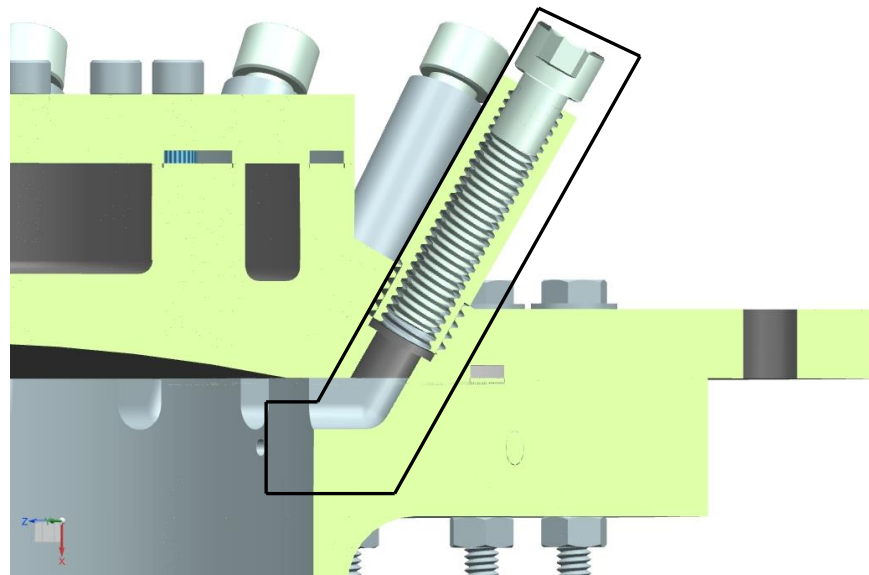
Figure 2.7 - CROME-X acoustic cavity

These tunable acoustic cavities are Helmholtz resonators and are designed to dampen out the 1T, 2T, and 3T modes which are all tangential modes. Resonant length calculations were

made for 1T, 2T, 3T, 1R, 2R, 1T1R, 1T2R at both 500 lbf and 2000 lbf thrust because CROME-X is a throttleable engine. Based on the calculations, the resonant lengths were similar at the two thrust levels so the resonant lengths can remain constant throughout the test campaign any may be adjusted based on need during testing. The acoustic cavities will be tuned by adjusting the distance the screw is threaded into the Inconel acoustic tube. The lengths will be adjusted to dampen out the 1T, 2T, and 3T modes because these are the modes most commonly seen. Figure 2.8 shows the placement of the needed resonant lengths to dampen out their corresponding mode. Table 2.3 shows the cavity number, the type of mode it is dampening, the cavity length needed (resonant length), and the resonant frequency.

Table 2.3 - Cavity lengths with corresponding number, mode, and frequency

Cavity Number	Mode Type	Cavity Length (in)	Resonant Frequency
1	1T	2.2	3516
2	2T	1.4	5428
3	1T - 300 HZ	2.3	3216
4	3T - 300 HZ	1.2	7010
5	1T	2.2	3516
6	1T + 300 HZ	2.2	3216
7	2T - 300 HZ	1.4	5128
8	3T	1.1	7310
9	1T - 300 HZ	2.3	3216
10	1T + 300 HZ	2.2	3816
11	2T + 300 HZ	1.4	5728
12	3T +300 HZ	1.1	7610
13	1T	2.2	3516
14	1T - 300 HZ	2.3	3216
15	1T + 300 HZ	2.2	3816
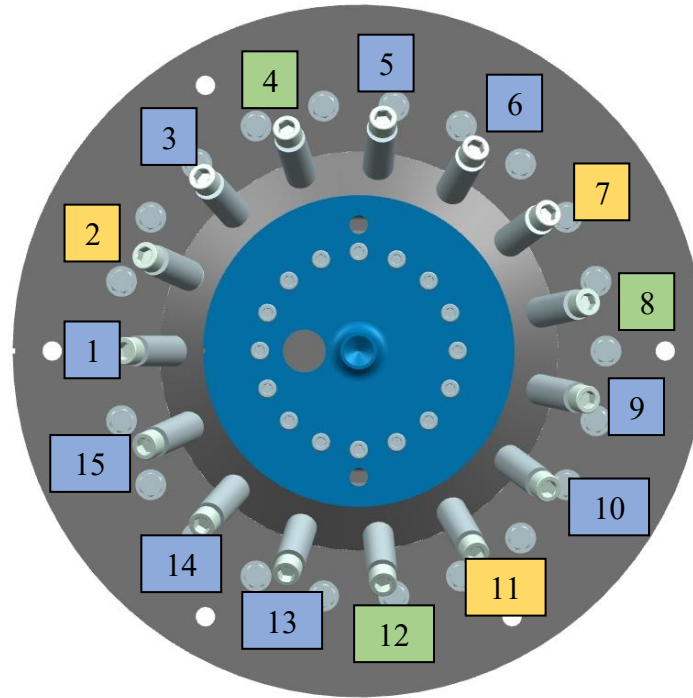
Figure 2.8 - Acoustic cavity resonant length placement

1T: blue | 2T: yellow | 3T: green

2.3 COMBUSTION CHAMBER

2.3.1 Overview

The combustion chamber is an Inconel 718 conical (15° half angle) chamber design and was designed for an optimal thrust of 1250 lbf because this thrust value lies between the desired 500 – 2000 lbf thrust range. If the chamber was designed to the lower end thrust, the nozzle would be under expanded for the higher thrust level and if the chamber was designed to the higher thrust level, the nozzle would be over expanded. This is the reason a median value was selected so that design is not overly under expanded or over expanded. For the desired thrust, a chamber pressure of 233 psi will be seen at 2000 lbf and 75 psi at 500 lbf. As mentioned earlier, the chamber utilizes a fuel film cooling method to cool the chamber. The design implements a contraction ratio (CR) of 4, an expansion ratio (ER) of 2.7, and a throat diameter (Dt) of 2.85 in. The current iteration of the chamber has a variable wall thickness which can be seen in Figure 2.9 - Combustion chamber cross-section of engine profile. Since this iteration of the chamber is

for initial testing and characterizing of the engine, a variable wall thickness will act as a heat sink and help cool the chamber and will allow the placement of 21 thermal couples placed along the chamber profile and at various angles around the chamber. This is to characterize the temperature gradient created during hot fire testing. There is also a 0.19" port on the chamber for mounting the ignitor.
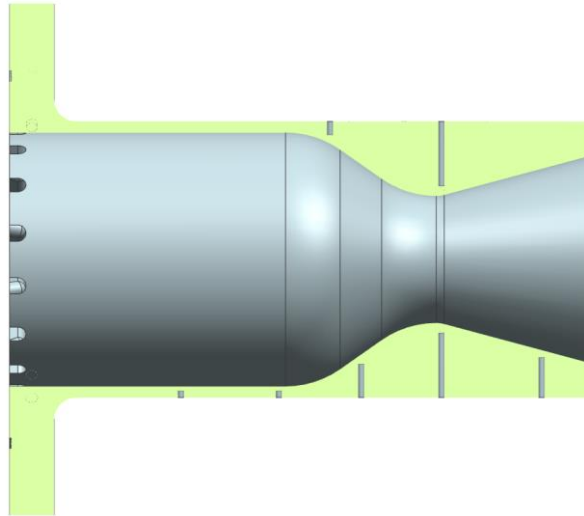
Figure 2.9 - Combustion chamber cross-section of engine profile

Table 2.4 - CROME-X Engine and combustion chamber parameters

Parameter	Value
Thrust Range	2000 – 500 lbf
Optimal Expansion Thrust	1250 lbf
Pc	233 – 75 psia
Dt	2.85 in
ER	2.7
Pe	6.3 (49% Pa)
Isp	~ 244 – 188 s
System Isp (includes 30% FFC)	~ 219 – 168 s
FFC Flow rate	0.95 – 0.3 lbm/s

Chapter 3: Simplified Engine Model

3.1 MODEL SYMMETRY

When creating a finite element analysis (FEA) model, a key motive is to reduce the run-time as much as possible. One method of doing this is via creating model symmetry. This means taking advantage of the design symmetry and only modeling a section of it rather than the entire design. This results in a reduced number of elements needed, which in turn decreases model run time. Model symmetry is performed on the 2,000 lbf engine, CROME-X, which has quarter symmetry in its design. For the simplified model, quarter symmetry will also be used to keep boundary conditions similar between the simplified model and the actual CROME-X model. Figure 3.1 shows the entire design with no symmetry reduction. This simplified model has a chamber, and injector, and a support. The chamber is marked in light blue, the injector is pink, and the support structure is grey.
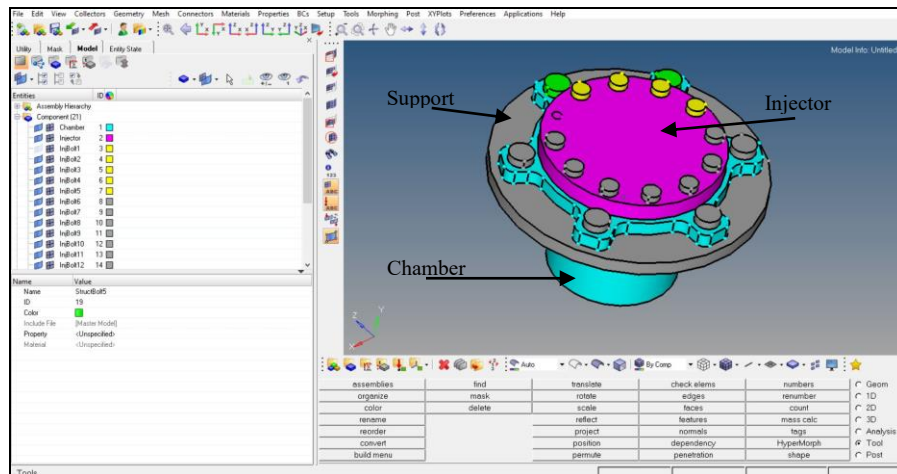
Figure 3.1 - Initial Import of Dummy Engine Design into Hypermesh.

Before importing geometry, ensure that the design is simplified in the CAD software. Remove any round, small holes such as film cooling holes that require electron beam additive manufacturing (EBM), and threads. All these features make the meshing process more difficult because these features affect the shape of the element. The mesh process itself will be described

later. Creating quarter symmetry from the full geometry can be done via Hypermesh. The process is as follows:

1. Select "Geom" located at the bottom right hand corner

2. Select "Nodes" and input x: 0, y: 0, and z: 0. This location should be the center axis of the model. If it is not, then this depends on how the engine design was created in the CAD software. Adjust the values to place the node above the model and 'reject' the node if it is not. Once the node is above the model, 'create' the node.

3. Create two more nodes such that the nodes encompass the desired quarter section of the model by adjusting the values. First adjust the x- axis until the node is in the desired location and then do the same in the z-axis. The analyst should note that the axis specification may be changed based on how the engine assembly is generated. In this simplified model, the datum plane can be seen in Figure 3.2.
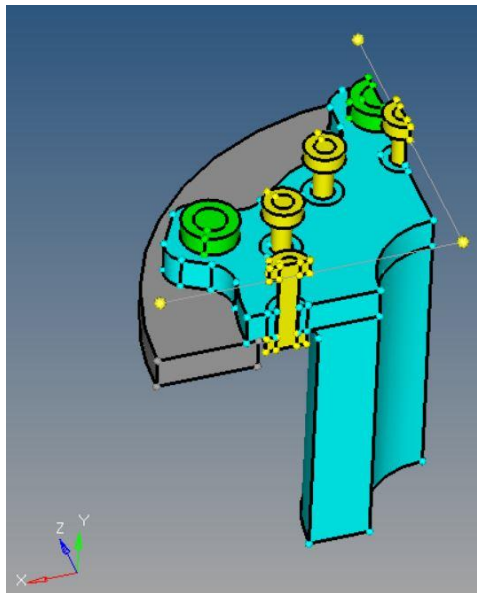
Figure 3.2 - Node positioning for creating symmetry.

4. Press 'return' and select "lines". Select the second icon from the left, "Linear Nodes". "Node Path" should be highlighted. If it is not, select it. First, select the center node using right click and select only one of the two remaining nodes. Once these nodes

15

are selected, press "create". A line connecting the nodes should have been created. Do this same process with the center node and the remaining node. Press 'return' when done.

5. Still in the "Geom", select 'solid edit'. Select the "trim with lines" option. The model has components listed under "Components". This is where the components can be color coded. Each of these components is its own solid.

6. Under 'with sweep lines' in the 'sweep to:' section, select 'by vector', 'y-axis' (this axis is perpendicular to the lines that were drawn in the step 5), and 'sweep all'. Once this is highlighted, select the chamber, the injector, and the support. Press 'solids' and once it is highlighted, select the injector, the chamber, and the support on the main screen. When these solids are selected, they should turn a white color. Now, press 'lines' and select one of the two lines. This can be seen in Figure 3.3 below. Once one of the lines is selected, press 'trim'. If there was an error in the trim, there should be an error message at the bottom right corner of the Hypermesh screen. When the solid is trimmed, the trimmed section will turn yellow as seen in Figure 3.4 below. Do this same process with the remaining solids. Solid edit using trim with plane/surf may also be used. It all depends on what method is easiest when dividing up and editing the solids into smaller solids.
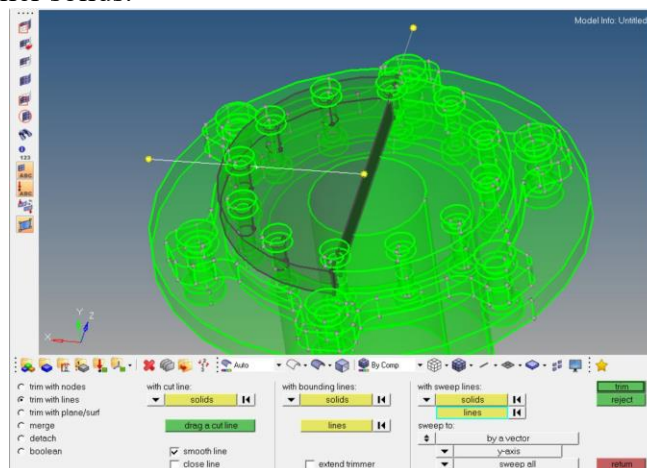
Figure 3.3 - Preparing to Trim Solids with Lines
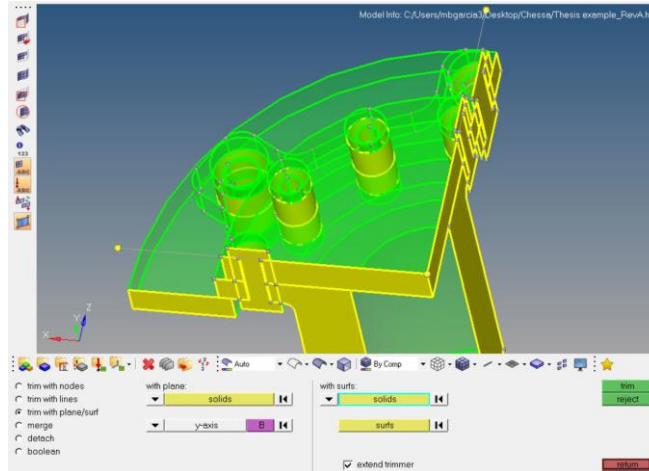
Figure 3.4 - Trimmed Solids.

7. To remove the unwanted solids from the model, right click in 'components' and select 'create', 'component', and name the component "unwanted solids". It is optional to change this component color to red. This helps monitor which solids are now in this component because the solid will change to red in the view screen.

8. This step is to reorganize the solids into other collectors. Begin by selecting 'tool' and then pressing 'organize'. Click 'solids' and then click the solids on the screen that need to be moved to "unwanted solids". Again, the solid should turn white when selected. In the 'dest component', select "unwanted solids". Once all desired solids are selected, press 'move'. All the selected solids should now be red.

9. Hide these solids by clicking the blue flag to the left of the component name or delete this component by right clicking "unwanted solids".

3.2 MATERIAL PROPERTIES

3.2.1 Material Card

In this analysis, the materials being used are Inconel 718 and ANSI 4130 steel. The Inconel is applied to the injector body, the chamber, and the bolts. The steel is applied to the support ring. The material properties are as follows:

Table 3.1 Material Properties Used in Simplified Model

	Conductivity (mW/mm-K)	Specific Heat (N-mm/K)	Density (kg/mm³)	Modulus of Elasticity (N/mm²)	Poisson's Ratio	Coefficient of Thermal Expansion
Inconel 718	6.5	4.35e-8	8.22e-10	158579	.28	1.2e-5
ANSI 4130 steel	6.5	5.05e-8	7.85e-10	205000	.28	1.3e-5

In addition to these materials, the actual CROME-X model will include Inconel 625. Inconel 625 is used in the injector section of the engine and Inconel 718 is only used for the chamber. To simplify analysis, only Inconel 718 will be used for all engine components in the simplified model.

When inputting the material properties, be mindful of units. No actual units are used in Abaqus nor Hypermesh, the analysis programs, so it is the analyst's job to monitor the units and ensure they match. For example, the geometry of the engine in this simplified model is in mm because it was designed in CAD software using metric units. This means that the corresponding units are N, mm, sec, and K.

To input the material properties into Hypermesh, click the 'Materials' tab and select 'create'. In this model, input the material name double check that the 'Card Image' is "ABAQUS_MATERIAL". If the Card Image does not read that, then the preference needs to be changed to Abaqus.

In the material card, check mark conductivity, specific heat, density, elastic, and expansion. For each of these sections, click on the table icon. Input the value in the first column and the corresponding reference temperature in the second column. The first column should be labeled either "k" for conductivity, "sp_heat" for specific heat, "E" for Young's Modulus of Elasticity, "Nu" for Poisson's ratio, and "alpha" for the coefficient of thermal expansion (CTE).

3.2.2 Property Card

The property card defines the material properties and card image for the components. Each component must be assigned a property for the analysis to run. To create a property card, click the 'Properties' tab, select 'create', then 'properties'. The window below should appear below in Figure 3.5.
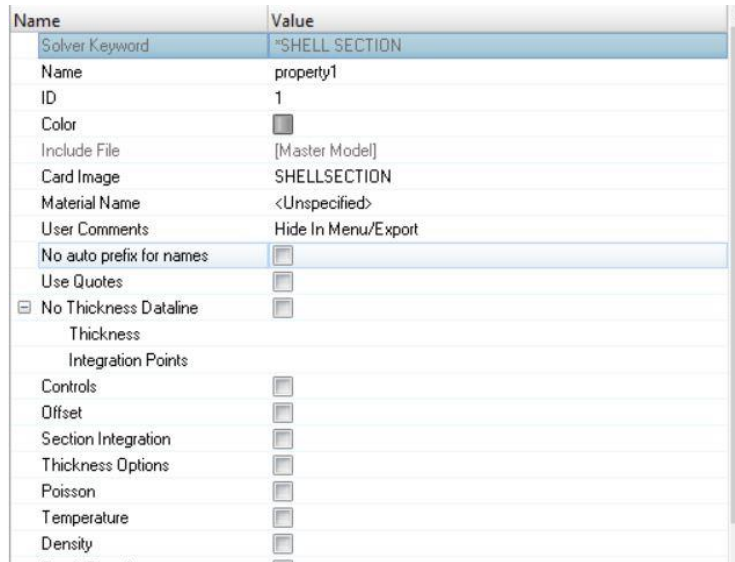
Figure 3.5 - Hypermesh Property Card.

For this model two properties were created, Inconel and Steel. The name should be changed to "Inconel" and "Steel", respectively, and the Card Image for both need to be changed from "SHELLSECTION" to "SOLIDSECTION" because the components are solids. The Material Name was also changed to its respective material.

The next step is assigning the property to the components. To assign the property, click on one of the components in the Model-Entities screen. In the section directly below, the property and material are displayed. It is in this window that the properties can be assigned as seen in Figure 3.6 below.
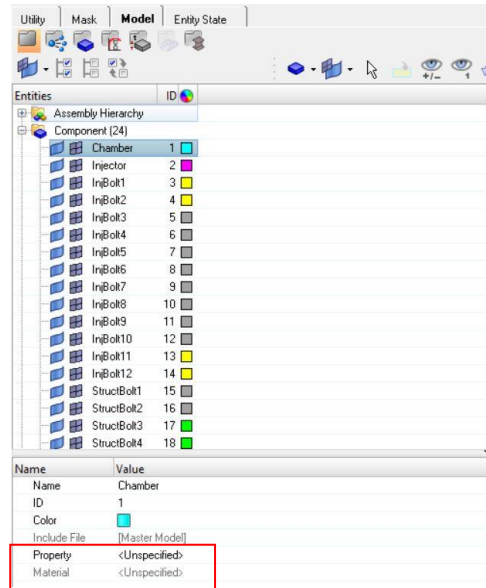
Figure 3.6 - Assigning the Property to the Component

3.3 MESHING

Utilizing solid edits is a tool that helps to ease the meshing process. Similar to what was done to create the quarter symmetry in section 2.1, for meshing, the solid edit breaks up more complex solids into a geometry that is easier to mesh. In Hypermesh, there is an option in '3D' called "solid map" that will be used to mesh the engine. The goal is to use mostly hexahedron elements in the model. A hexahedron is defined as any polyhedron that has 6 faces. The hexahedron element has a node at each of the 8 vertices which categorizes the element as first order. If the element was second order, there would be an additional node that is equidistant between any two nodes as seen in Figure 3.7. For the second order hexahedron, there are 20 nodes in total. For this analysis, first order elements will suffice.
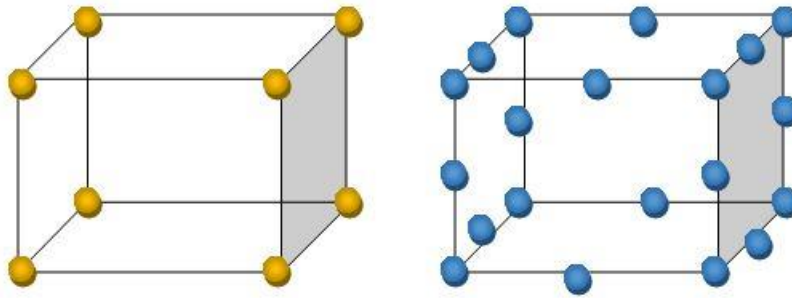
Figure 3.7 - First and Second Order Hexahedron Elements

The method to meshing the model is to use "solid mesh". In solid mesh, a solid is selected as well as starting surface and a destination surface. These two surfaces must be similar for the solid mesh to work, otherwise, the mapped meshing will fail. Be sure to use first order hexahedrons in all areas where there is contact and for the bolts. The first order element is the default so there is no need to select an option to ensure the elements are first order. The hexahedron is one element type that works with contact interaction. If there is one solid section that cannot be solid mapped meshed and is not in a location of contact, the volume tetramesh option can be used. Again, try to use hexahedrons first before using the tetramesh. For this simplified model, an element size of 1 was used throughout the entire model.

3.4 CONTACTS

A contact interaction card simply defines a contact interface between two components. Although the definition is simple, getting the FEA model with contacts to run and converge is a bit more challenging.

To begin, creating contacts for the model is simpler in Hypermesh compared to Abaqus, therefore; Hypermesh was used for contacts. The process of creating contacts is explained in the following steps:

1. Opening Contact Manager - Change the window to "Utility". The 'Contact Manager' can be found near the bottom of the screen under "Tools". This can be seen in the image below.

2. Setting up Inputs to Locate Contacts - The Abaqus Contact Manager window should pop up. There, click 'Auto' as seen in Figure 3.8. Now the Auto Contact window should have popped up. For the proximity distance, input a value less than the radial bolt hole clearance. This is to ensure that the auto-contacting does not take the bolt and the clearance as in contact. The value used in this analysis was 0.005. In the interaction section, create a new interaction named, "Interaction1". For the Surface Interaction window, leave all values as default because these values will be set in Abaqus. Once everything in inputted, go ahead and press 'okay'.
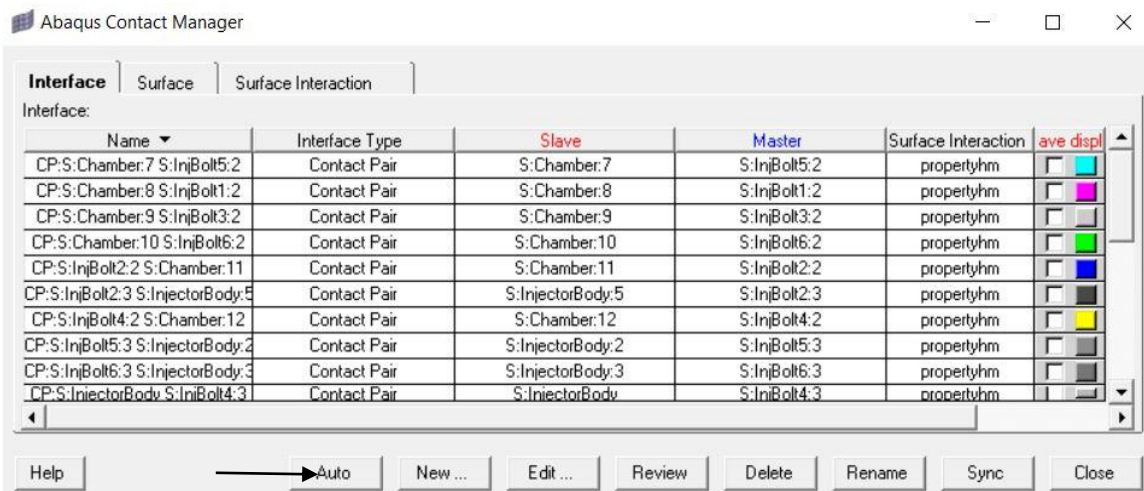
Figure 3.8 - Auto Contact Window

3. Selecting and Verifying Contacts - Back in the Auto Contact window, select two components to find the contacts for. Press 'Components', click 'comps', and select the desired components. A good approach to creating contact interactions is to select two components that are in contact at a time. In the figures below, the injector and bolt were used. Once the components are check marked, press 'select' and 'proceed'. The selected components should now be listed under 'Components' in the Auto Contact window. Then click 'Find'.
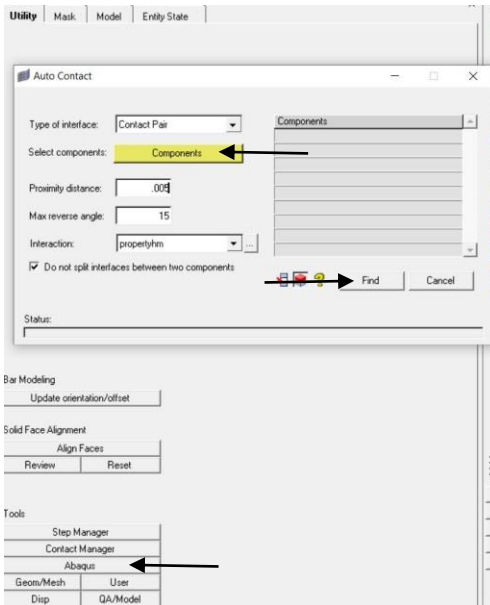
Figure 3.9 - Creating Contacts Input Window

4. Editing Contact Surface - The auto contact automatically creates a contact between the selected components. These contact interactions usually need to be edited. In the AutoContact manager, right click one of the components that listed under 'Interfaces'. Then select 'edit faces'. Use the right click button on a mouse to deselect elements that are out of the contact range. Press 'proceed' when the editing is complete. The figure below shows a contact surface that needs to be edited because there is an element highlighted in white that is not in the desired contact area. In cases like this, the 'edit face' option should be used to remove any undesired elements.
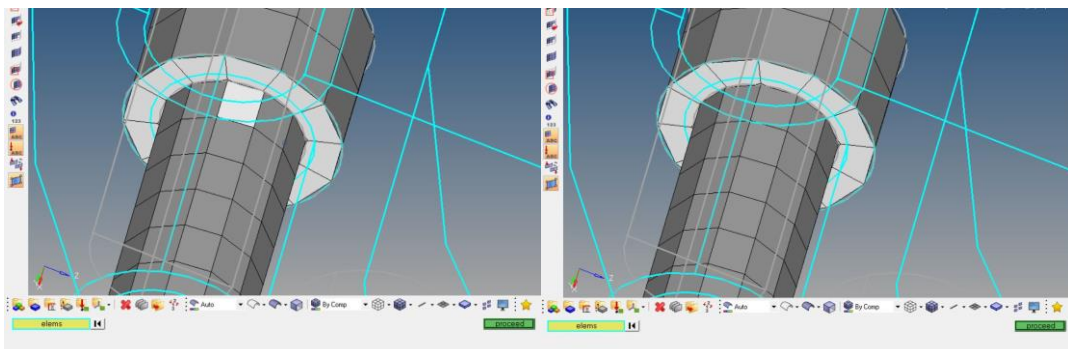
Figure 3.10 - Unedited Contact Surface versus an Edited Contact Surface

5. Completion of a Contact Surface - Once the contact has been created and edited, click 'Create' on the bottom left side of the screen. This will create the contacts. Repeat this process for all components that have contact.

When all the contacts have been created, the result should look as seen in Figure 3.11. Be sure to check that the selected elements matchup between the slave and master surfaces and that there are no selected elements that are beyond that area. If there is, remove those elements from the selection by following step 4.
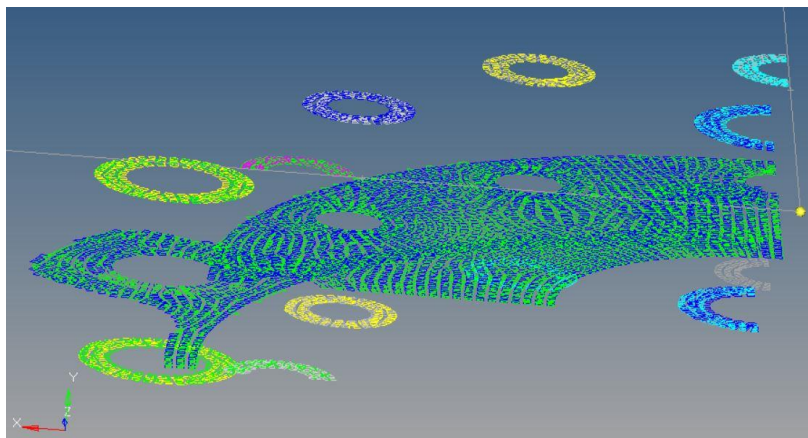
Figure 3.11 - Contacts Created in Hypermesh

The Hypermesh model can now be exported to continue creating the model in Abaqus. The export options can be found in the File drop down menu and select the Solver Deck option. Check that the File Type is Abaqus and select the desired location of the file in the File line. Once all these options are selected, press Export to export the model as an .inp file.

Chapter 4: Mechanical and Thermal Analysis – Hypermesh

4.1 MESH

So that there can be a model comparison between two modeling softwares, an analysis for the CROME-X will be created in Hypermesh where hexahedron elements be used as well. A base element size of 0.125 will be used for all components in the model and in this analysis, the chamber, the injector body, and the pintle manifold will be analyzed separately. The Preference in this model must be changed to Optistruct before the meshing process begins because if changed later, it will affect the elements and generate an error whilst running the model. Again, this is to provide a comparison for results obtained in the Abaqus model of the CROME-X engine assembly. This section provides an explanation of the methodology used to mesh each component.

4.1.1 Chamber

The solid edit procedure is the same as the procedure mentioned in section 3.1. The main idea here is to divide the component into multiple smaller solids that are easier to mesh, meaning that the beginning surface and end surface of the solid are similar. For example, in Figure 4.1, the area within the yellow line was one solid before the solid edit. The solid mesh option would not be able to be used for this section, so the solid must still be divided into smaller solids as seen will the solids marked by the red lines. For the bolt holes, use the bolt head surface to edit the solid as seen by the blue marks below.
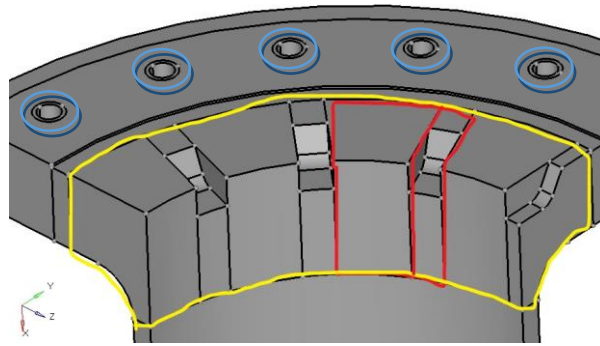
Figure 4.1 - Combustion Chamber Solid Edit

When beginning the meshing process, it is best to start map meshing the more difficult geometry or the geometry that a load will be applied to. The bolt hole area is an area that should be started with and the surrounding solid meshes will be dependent of the bolt mesh because a bolt preload will be applied to this area. There should be no issues using the Solid Map > Multi Solids option to mesh these bolt hole regions. Just make sure to change the Source Shells to 'quads' and check mark the "stop meshing on bad Jacobian" options and change the Elem Size to 0.125. Figure 4.2 shows how the solid map process looks for the bolt hole region.
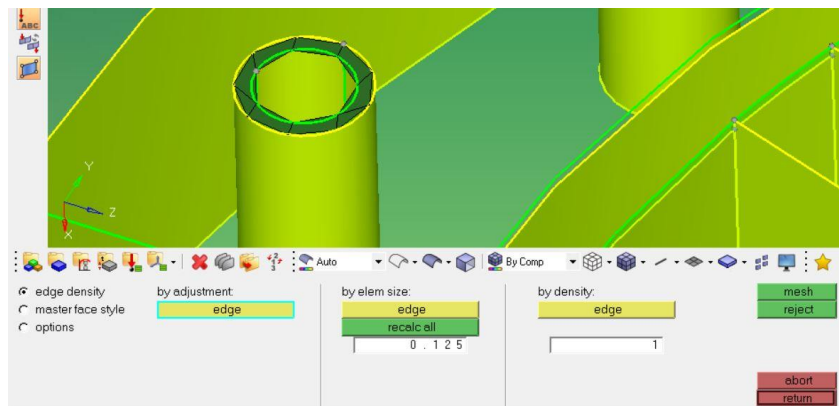
Figure 4.2 - Not Adjusted Mesh

In this figure, the current mesh is not the idea type of mesh that the analyst desires. The mesh density must be adjusted until the mesh is as desired. Either the By Adjustment or the By Density option may be used. Typically for this cylindrical solid, having the nodes on the outer diameter and inner diameter equal will create the desired mesh. In Figure 4.3, the adjusted mesh for this bolt hole region can be seen. The number of nodes on the inner and outer diameters is 12 and the elements created are aligned and are rectangular in shape, which is what is desired.

Figure 4.3 - Mesh Adjustment for Bolt Hole Region

The acoustic cavity region of the combustion chamber is one of the most difficult areas to mesh due to its complex geometry. To assist in the meshing, the first step is to divide this region up into easier solid mappable regions as described earlier in this section. This section of the chamber requires multiple steps for meshing and is not simply using the Multi Solids options. To begin, the 2D mesh option will be used. Create a new component labeled "2D" and change the color so that these elements are distinguishable in the model. Th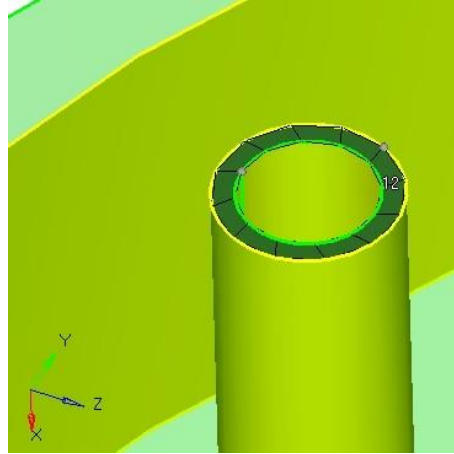is new component is needed to group all the 2D elements in one place or component so they can be deleted later one. The 2D elements can be accessed by clicking 2D > automesh. Change the Element Size to 0.125, change the Mesh Type to 'quads', and change from "elems to surf comp" to "elems to current comp". The last change is to make sure 2D elements are grouped to the 2D component that was created and not to its surface component, the chamber. As seen in Figure 4.4, two components for 2D elements were created as depicted by the yellow and green elements. This is because there are two different surface regions that needed to be meshed. When selecting the surfaces for meshing, select all the green region surfaces (surfaces with green elements in the figure) and click Mesh. Doing this should ensure that the surfaces will all have the same number of elements.
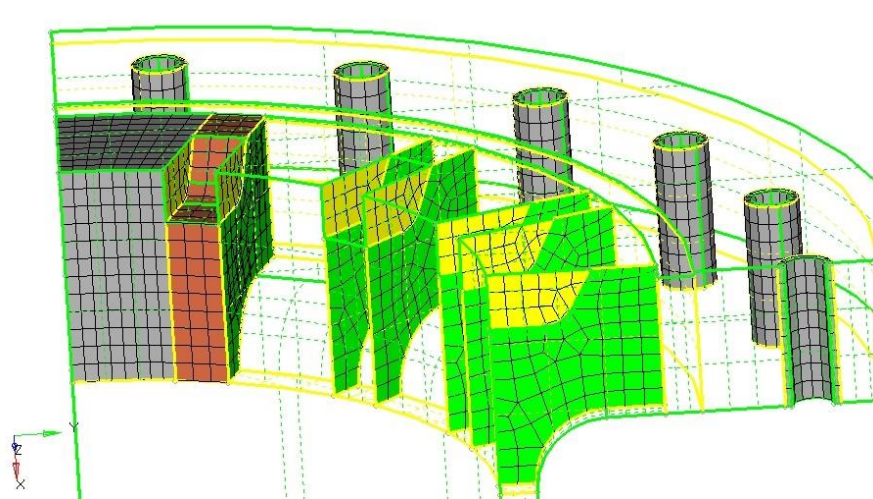
Figure 4.4 - 2D Elements used to mesh the acoustic cavity region

The region designated by the grey colored elements can be meshed using the 3D > Solid Map > Multi Solids option. Solids after this point must be meshed using the 3D > Solid Map > Line Drag option. The elements that should be selected in the Elem to Drag are the elements that are in the solid before. The Along Geom lines should be all the lines that are connecting the beginning surface to the ending surface. If the mesh was successful, the generated elements will be a dark orange color and a new component called "solid map" will be automatically generated. The analyst should note that if any more elements are created using this method, all these future elements will be grouped into the "solid map" component. If there are errors, it is beneficial to reorganize the newly created "solid map" elements into a new component. The different components created for this purpose can be seen in Figure 4.5. The various colored elements represent the newly created components to house the elements. The elements can be moved by selecting Tool > organize > click elements > by collector > check mark "solid map" component > click Select > Dest Component = "name of new component created to house generated elements" > click Move. If the elements were moved successfully, the element color on the screen should change to the new element color. Once all the solids in the acoustic cavity region have been created, the model should look like the figure below.
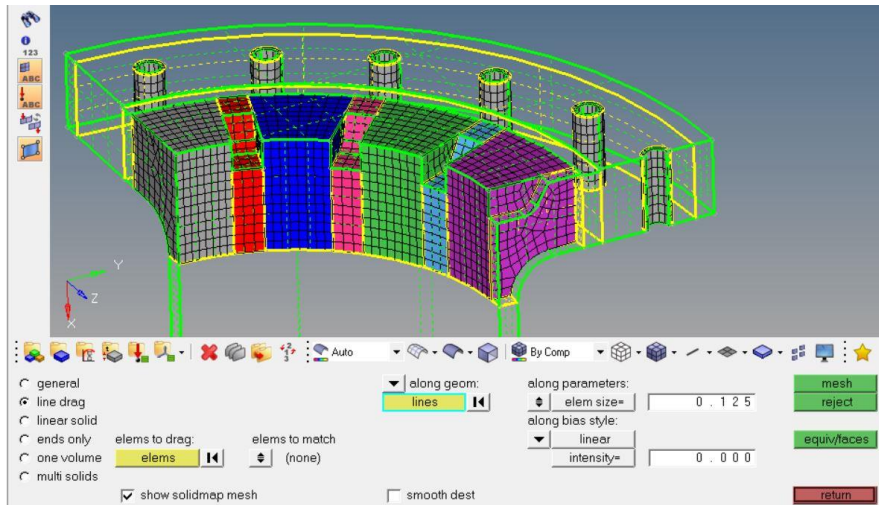
Figure 4.5 - Successful meshing using 2D elements and 3D Line Drag

Meshing the remaining combustion chamber solids can be completed using both the Solid Map > Multi Solids method and the 2D element with the 3D Line Drag method. Once all these elements have been generated, an equivalence in needed. Equivalence is needed when there are two or more nodes located in the same location of the mesh. This usually occurs the Multi Solids is not used when creating elements for neighboring solids. Equivalencing the model can be done by selecting Mesh on the tool bar > Check > Nodes > Equivalence. For the components, select the Chamber component because this is the current working component and then select Preview Equiv. If there are nodes that need equivalencing, the nodes will be highlighted on the screen. After previewing, click Equivalence and the nodes are now equivalenced. After this step, the elements can now be reorganized back to the Chamber component. This is like the reorganization that was done earlier and if the elements were successfully transferred, the elements on the screen should change to the designated component color. The total number of nodes and elements in the Chamber component can be seen in the figure below.
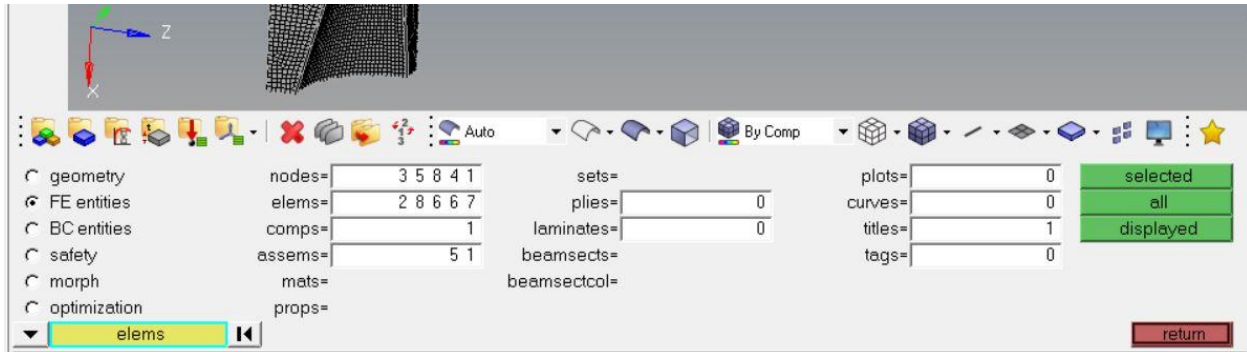
Figure 4.6 - Total number of elements and nodes in the chamber mesh

4.1.2 Injector

The solid edit procedure is the same as the procedure mentioned in section 2.1 and the combustion chamber solid edit methodology. Figure 4.7 shows how the injector body component was divided into multiple solids for mapped meshing. Again, the bolt head surface for all bolt connections was used to create another solid as depicted by the black circle on one of the injector bolts. For the injector, the same mesh density adjustment was used. In the initial solid map mesh of the bolt hole region, the inner and outer nodes were different values so that needed to be changed so that they are equal. The image on the far right in Figure 4.7 shows the mesh post mesh density adjustment. The inner and outer node values were both 13 in this case. This value may change for each bolt area and that is perfectly fine, the goal is to obtain a mesh that has all rectangular elements.
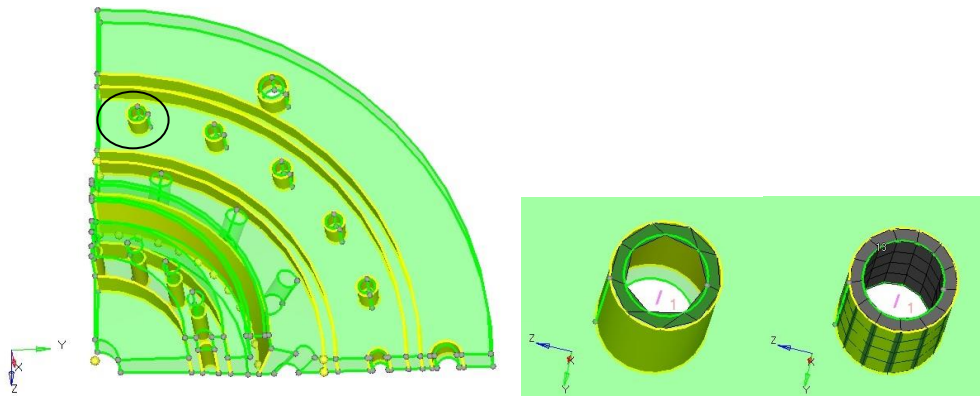

Figure 4.7 - CROME-X Injector solid edit cuts and mesh density adjustment

For the injector body component, the Solid Map > Multi Solids method for most of the solids. The injector geometry is less complex compared to the combustion chamber due to the acoustic cavities. The injector body does have acoustic cavities as well and that area is the most difficult to mesh. All the dark gray elements in Figure 4.8, were generated using Mulit Solids which is makes up the majority of the elements. The pink elements were generated using the 2D quad automesh/3D Line Drag method. The overview of how to achieve this mesh is described in section 4.1.1. The turquoise elements were generated using the 3D > tetramesh option. As mentioned earlier, the tetramesh is the last option element type when trying to achieve a hexahedron mesh. The tetramesh will still work but keep the use of this element type limited. To access this option, select 3D > tetramesh > volume tetra > click solids and select desired solid to mesh > change 2D type to quads, 3D type to mixed, and Element size to 0.125 to keep element size throughout the model consistent.
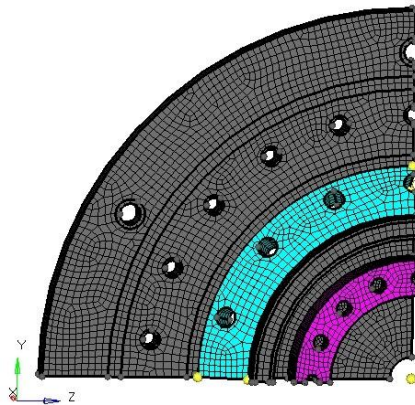
Figure 4.8 - Various meshing methods used to mesh the injector body

4.1.3 Pintle Manifold

The pintle manifold uses the same solid edit procedure and was divided into multiple solids for mapped meshing. In the initial solid map mesh of the bolt hole region, the same mesh density adjustment as the chamber and injector body was used. Figure 4.9 shows both the solids that the pintle manifold was divided into and the mesh that was generated. The entirety of the

pintle manifold mesh was meshed using the Solid Map method. The element size used for the
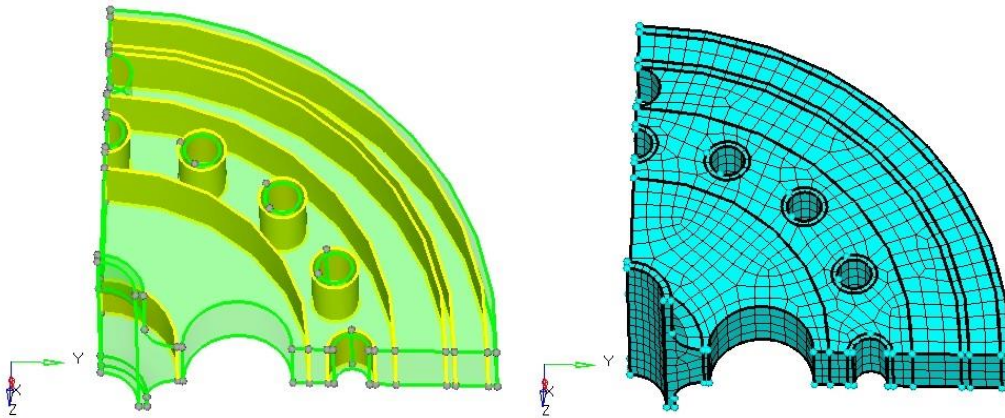
pintle manifold mesh was 0.125.

Figure 4.9 - CROME-X pintle manifold solid edit and mesh

4.1.4 Properties

In this analysis, the materials being used are Inconel 625, Inconel 718, and stainless steel 304. Inconel 625 is applied to the injector body and the pintle manifold, Inconel 718 is applied to the chamber, and the steel is applied to the support ring, the bolts, and the screws. The material properties from Special Metals are as follows:

Table 4.1 Material Properties at various temperatures [6][7]

Material	Temperature (°F)	Conductivity (lbf/s-°F)	Density (slinch/in³)	Young's Modulus (psi)	Poisson's Ratio	Coefficient of Thermal Expansion (in/in-°F)	Yield Strength (ksi)
Inconel 718	70	1.39	0.00077	29E6	0.29	7.31E-6	170
Inconel 718	1000	2.43	0.00076	24.8E6	0.271	8.09E-6	155
Inconel 718	1600	3.33	0.00076	20.0E6	0.331	8.91E-6	35
Inconel 625	70	1.22	0.00079	30.1E6	0.278	7.1E-6	70
Inconel 625	1000	2.18	0.00079	25.9E6	0.305	7.8E-6	60
Inconel 625	1600	2.85	0.00079	21.4E6	0.336	8.8E-6	30
Stainless Steel 304	70	1.8	0.00074	290E6	0.29	9.61E-6	31.2

The material property cards, and the corresponding property cards are creating using the same process as 3.2 Material Properties. The 1700°F is the adiabatic is flame temperature reached in the combustion reaction between oxygen and methane at the design MR. The 70°F are the material properties at room temperature. There was initial analysis done on an earlier iteration of CROME-X using the 1700°F wall temperature and the injector body did not meet the

required safety factor. This was due to the large temperature gradient created between the chamber well temperature and the cryogenic propellant temperatures. For this analysis, material properties were taken at 1000°F because the combustion chamber wall temperature is not expected to reach the 1700°F and the 1000°F still is slightly on the higher end. The property type for this model will be Solid Section. There should be a total of three material cards and three property cards. Once these property cards are created and are linked to its respective material card, the properties can be assigned to the components.

The yield strength values were obtained from the Special Metals Corporation and can be seen in the figures below. As the temperature increases, the yield strengths drastically decrease and plateau after a certain temperature. The yield strength values that will be used for this analysis are the yield strengths at 1000 °F.
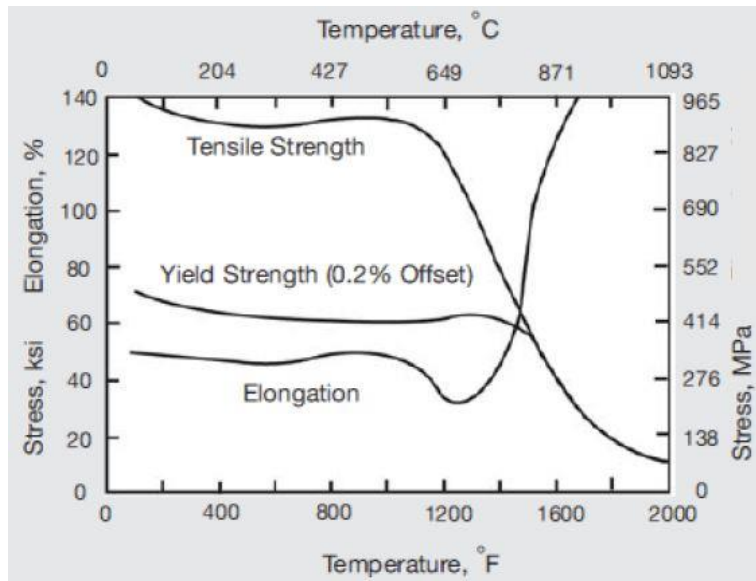

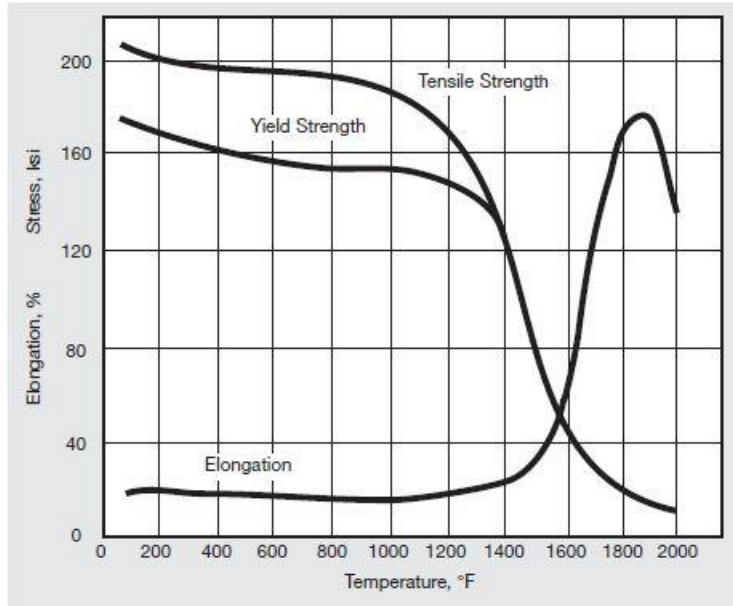Figure 4.10 - Inconel 625 yield strength [6]

Figure 4.11 - Inconel 718 yield strengths [7]

4.2 BOUNDARY CONDITIONS

The CROME-X has a combination of single point constraints (SPC) and mechanical and thermals loads. The SPCs are used to limit or constrict degree of freedoms (dof) of the components. The dof that can be constrained are the 1, 2, 3, 4, 5, and 6. The 1-3 dof are for displacement and the 4-6 are for rotation. Because a quarter symmetry analysis of the CROME-X is being conducted, there must be symmetry constraints applied to the model to represent to the model that there is still more of the component that is not in the FEA model. The three degrees of freedoms that are constrained to model a symmetry boundary condition are the displacement dof (x-, y-, z-axis) that is perpendicular to the surface and two rotation dof are the rotation dofs that are not on the same axis as the displacement dof. For example, as seen in Figure 4.12 - Creating a symmetry boundary condition, the displacement dof that is perpendicular to the surface is the y-axis (2). Therefore, the rotation dof that need to be constrained are the x- and z-axis rotations (4, 6). A vertical constraint, or in this case, an x-direction constraint because of the injector component that is mounted on top of the chamber flange.
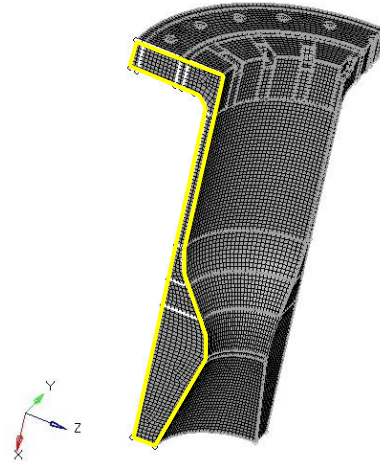
34

Figure 4.12 - Creating a symmetry boundary condition

The symmetry boundary conditions, mechanical loads, and thermal loads that are applied in the model can be seen in the tables below. The explanation on creating these loads are explained in later sections.

Table 4.2 - Hypermesh chamber mechanical boundary conditions

Single Point Constraints (SPCs)	Loads
- Quarter Symmetry - SPC's - Y-sym: 2,4,6 - Z-sym: 3,4,5 - X-direction: 1	- Chamber Pressure - Chamber Bolt $A = \frac{\pi}{4}(\frac{7}{16})^2$ (based on bolt head diameter) $P = \frac{F\,(Preload)}{A}$

Table 4.3 - Hypermesh chamber thermal boundary conditions

Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
    ○ Y-sym: 2,4,6
    ○ Z-sym: 3,4,5
    ○ X-direction: 1
	• Chamber Temperature
	• Acoustic Cavity Temperature
	    ○ 80% of chamber temperature
	• Convection to ambient

Table 4.4 - Hypermesh injector mechanical boundary conditions
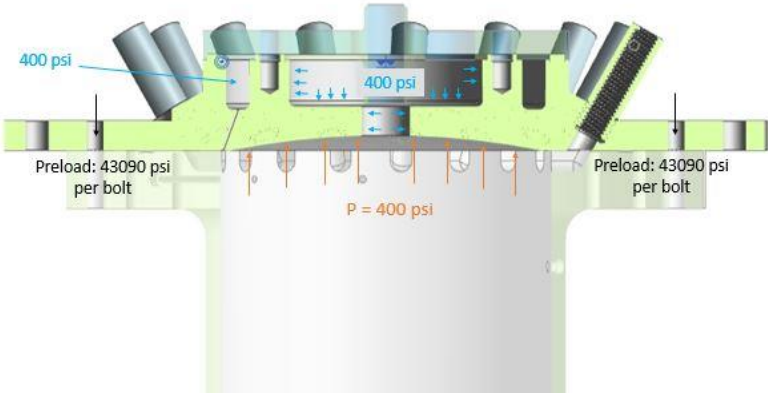
Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
    ○ Y-sym: 2,4,6
    ○ Z-sym: 3,4,5
    ○ X-direction: 1
	• Chamber Pressure
	• Propellent Pressure
	• Chamber Bolt
	    ○ $A = \frac{\pi}{4}\left(\frac{7}{16}\right)^2$ (based on bolt head diameter)
	    ○ $P = \frac{F\,(Preload)}{A}$

Table 4.5 - Hypermesh injector thermal boundary conditions
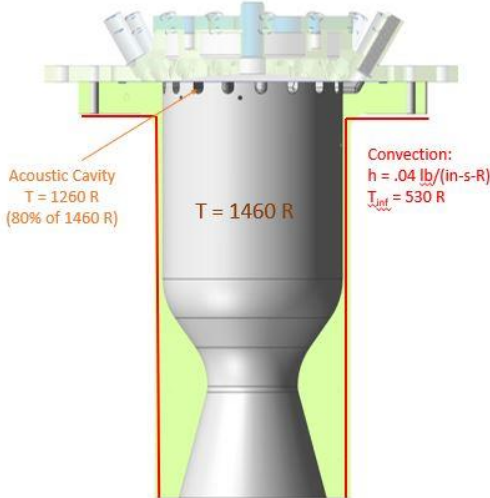

Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
   ○ Y-sym: 2,4,6
   ○ Z-sym: 3,4,5
   ○ X-direction: 1
	• Fuel Temperature
	• Oxidizer Temperature
	• Acoustic Cavity Temperature
	   ○ 80% of chamber temperature
	• Convection from chamber temperature
	• Convection to ambient

Table 4.6 - Hypermesh pintle manifold mechanical boundary conditions

Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
   ○ Y-sym: 2,4,6
   ○ Z-sym: 3,4,5
   ○ X-direction: 1
	• Propellant Pressure
	• Injector Bolt
	   ○ $A = \frac{\pi}{4}\left(\frac{7}{16}\right)^2$ (based on bolt head diameter)
	   ○ $P = \frac{F\,(Preload)}{A}$

Table 4.7 - Hypermesh pintle manifold thermal boundary conditions

Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
    o Y-sym: 2,4,6
    o Z-sym: 3,4,5
    o X-direction: 1
	• Fuel Temperature
	• Oxidizer Temperature
	• Convection to ambient

4.2.1 Mechanical

The mechanical loads include the chamber pressure that is created during the combustion process, the pressure created by the cryogenic propellants once they enter the injector via the facility lines, and the bolt and screw preloads that are inputted to ensure that the seals within the engine assembly are pressurized enough to seal properly so that there are no leaks through the engine components. The combustion chamber pressure boundary condition input reflects the maximum chamber pressure that the CROME-X can achieve. This maximum chamber pressure of 235 psia is for the 2000 lbf thrust level. The actual pressure value used in the analysis was 400 psia because 400 psia or 413 psig is the maximum controlled pressure that the facility can provide. The facility for both the oxygen side and the fuel side are blow down systems and the pressure achieved is driven by the maximum pressure the tanks can withstand. The 400 psia pressure is also applied to the fuel manifolds and the oxygen manifold in the injector because of the facility restrictions. The bolt and screw preloads as described in section 2.2.2 were applied as

pressures in the analysis. The pressure value was calculated by dividing the preload by the area of the bolt head or the screw head. This method of applying the bolt and screw preloads reveals one reason why the during the solid edit process, the bolt and screw hole areas are edited using the bolt head or the screw head. This cut creates the exact surface area to apply the load to. To create this load in Hypermesh, select Analysis > Pressures > change Node to Surf if the option is not already Surf. Once in this window, input the magnitude or value of the pressure and a negative sign may need to be added to this value if the load is not applied correctly, but this can be changed later. Also, select the surfaces where the load is applied and once all surfaces are selected, the load can now be created.

4.2.2 Thermal

The thermal loads include the chamber temperature that is created during the combustion process, sub-zero temperatures generated by the cryogenic propellants within the injector, and the convection that occurs from the injector face to the chamber and the convection from the thrust chamber assembly to ambient. To create a surface temperature, create a new constraint via creating a new loadcol. This can be done by right clicking below Load Collector and selecting Create, then selecting Load Collector. Then, go to the Analysis section and select Constraint. Once constraint is clicked, a similar window to the figure below should appear.
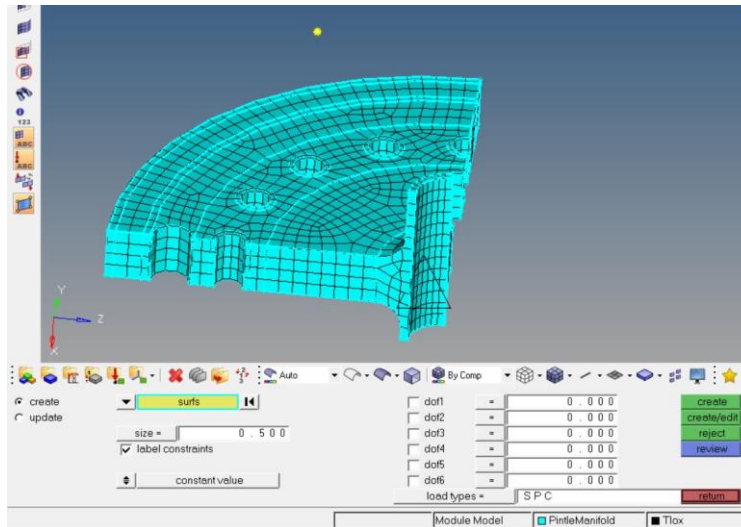
Figure 4.13 - Creating surface temperature constraint

Change the selection method from node to surfs so that the surfaces may selected. All the dofs should be unselected as well in this constraint. The constraint first must be created, and the card will be changed after to be able to apply a temperature value. As shown by the black arrow in Figure 4.14, select Card Edit and select one of the temperature constraints in the loads. In Config, select const and in type, select SPC. Once the load and the configuration, and the type have been changed, click edit.
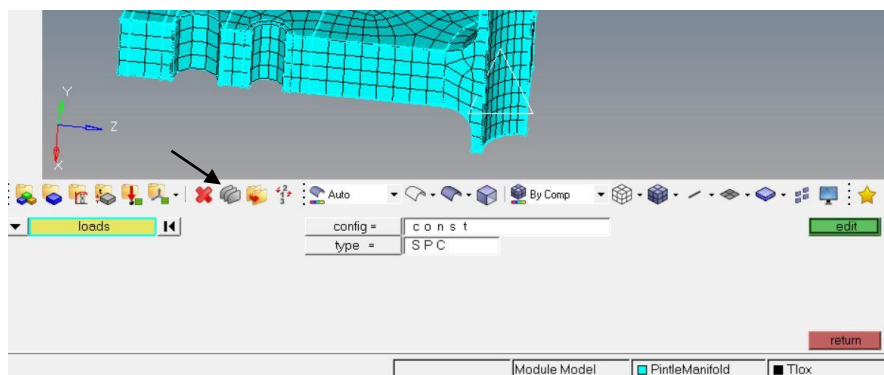

Figure 4.14 - Changing the settings to create temperature constraint

Once edit has been selected, the following window should appear. The temperature value in Rankin can be inputted into the box below D as seen in the figure below. This process will be repeated for each surface temperature that needs to be created in the model. After all the surface

temperatures have been created, they must all be reorganized and grouped into the same load collector. This is the same as the reorganization that was described earlier in this paper.

Figure 4.15 - Inputting temperature in the constraint

The convection interaction is created by selecting the BCs on the tool bar, then clicking Create, followed by clicking Interaction. When the window opens, change the Card Image to Convection and click 0 Elements to select the elements to apply to convection interaction. When selecting the elements, change the elems to faces, click Solid Elements and select All. When the All is selected, all the elements in the component should turn white. Then click on the Nodes > by geom > surf and select the surfaces to apply this interaction. When all nodes are selected, press the Add button as depicted by the black arrow in Figure 4.16.
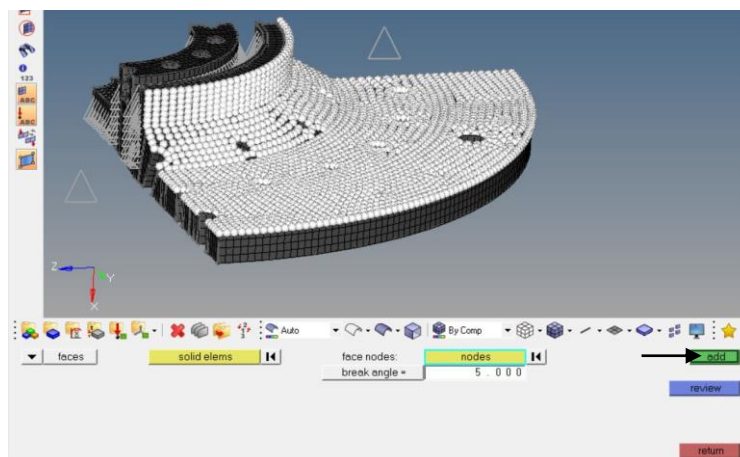

Figure 4.16 - Selecting elements and nodes for convection interaction

After the elements and nodes are selected, the create group window should reappear and the number of chosen elements should be shown in the Slave Entity IDs. In this case, the number of elements is 2210. A material also needs to be inputted into the MID section. The material

cards should have already been created in previous steps. It is a good idea to check if the material card includes a MAT4 card. The MAT4 card is where the heat transfer coefficient is inputted. This heat transfer coefficient is needed for the convection interaction to function. The heat transfer coefficients can be found in the boundary conditions tables.
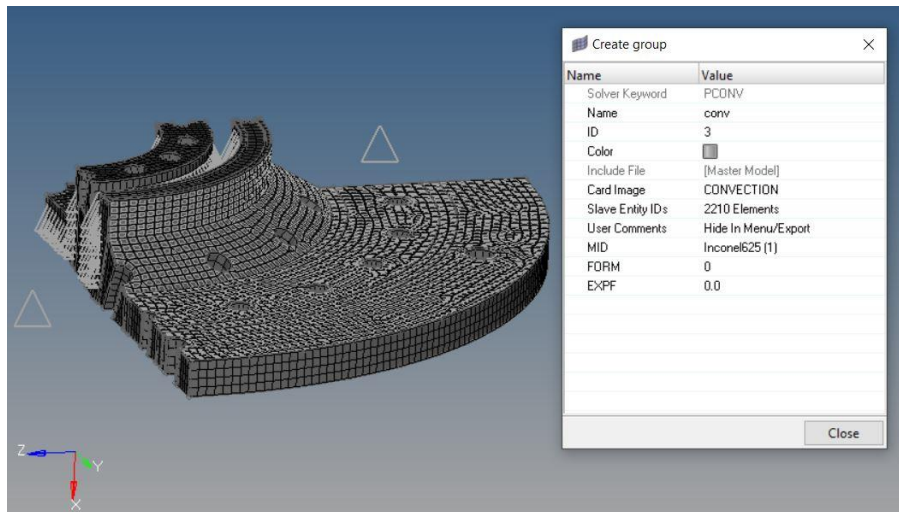

Figure 4.17 - Creating the convection interaction

The new load collector can now be assigned to the heat transfer load step. Create a new load step and change the analysis type to Heat Transfer (steady-state). The new temperature constraint load collector that was created to hold all the surface temperatures will be assigned to the SPC section. The rest of the of the sections in the Subcase Definition will be left unassigned. In Subcase Options, check ANALYSIS, OUTPUT, FLUX, and THERMAL. Within FLUX and THERMAL, click the icon and select H3D for the format.

After the heat transfer load step has been created, has ran, and the resulting temperature gradient has been checked to ensure that results are realistic and make sense according to engineering judgement, a thermal load step needs to be created. This is the step that will give results for the thermal stresses. For this load step, change the analysis type to Linear Static, in the SPCs select the symmetry and displacement load collector, have analysis check marked as well as the stress box within the Output section, and check mark Subcase Unsupported. In Subcase Unsupported, input "TEMP=2". This allows the load step to use the data that was generated in

the heat transfer load step. The number simply denotes the load step ID number so this may change from model to model.

Now that both the thermal and mechanical load steps are running and results are outputted, the combined load step can be created. The combined load step is a combination of the mechanical loads and the thermal loads. Creating this load step is nearly identical to the thermal load step. The only difference is the adding the mechanical loads to the step. This can be done by inputting the mechanical load collector from the mechanical load step into the Load section.

4.3 RESULTS

The goal for this analysis is to ensure that each component achieves a safety factor of 1.2 for both mechanical loads and thermal loads. The results for each component are listed in the Table 4.8 below. The yield strength values that will be used to determine the safety factor are in Table 4.1. To calculate the safety factor, simply divide the yield strength by the maximum stress that was seen in the analysis. The results of the mechanical analysis, thermal analysis, and the combined analysis for each component are shown in the figures below. The first three figures are the combustion chamber analysis, the next three are the pintle manifold analysis, and the final few figures are the injector body analysis.
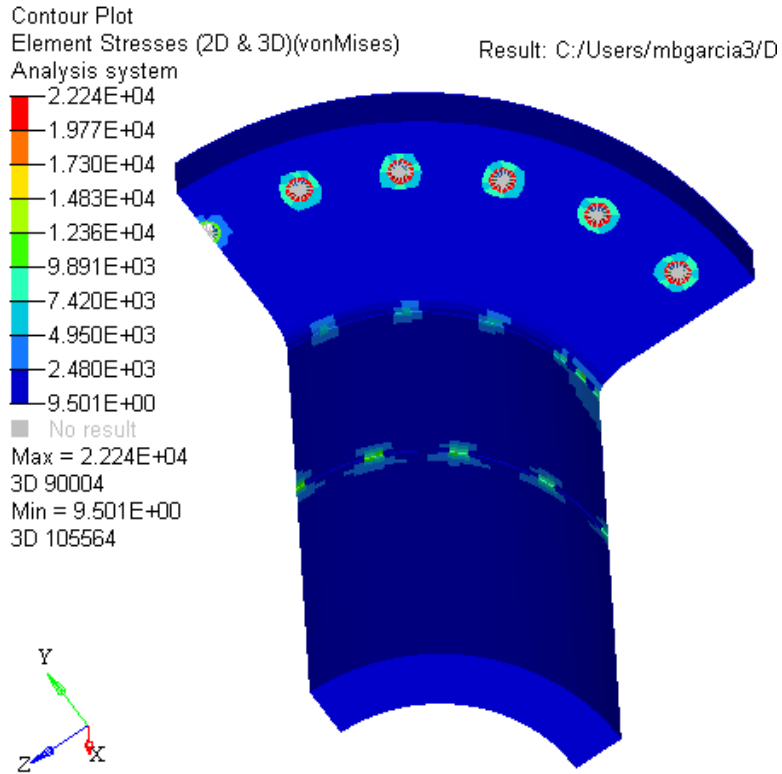
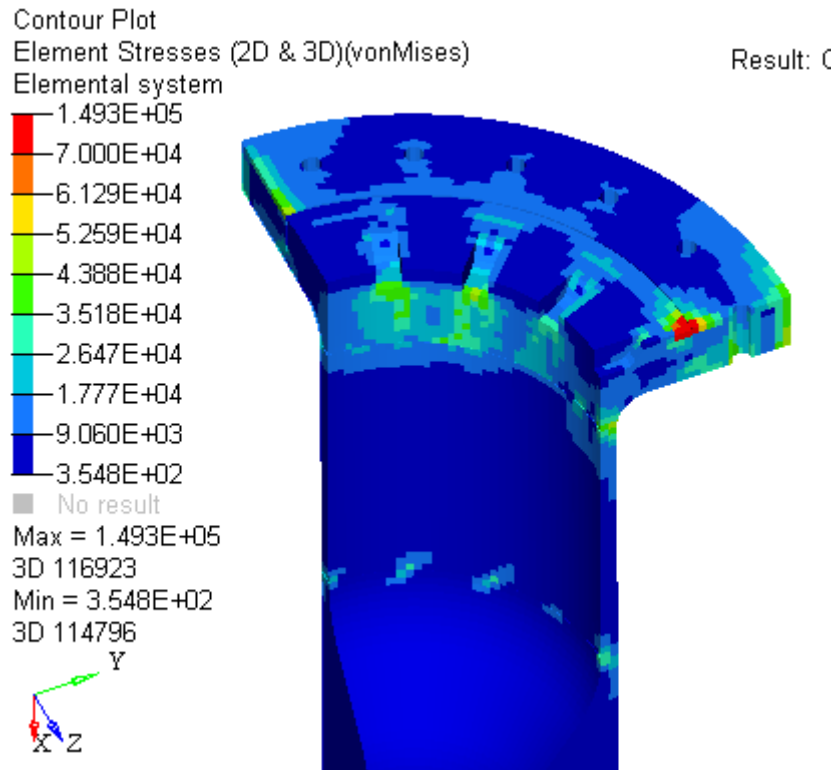Figure 4.18 - Hypermesh chamber mechanical stresses

Figure 4.19 - Hypermesh chamber thermal stresses
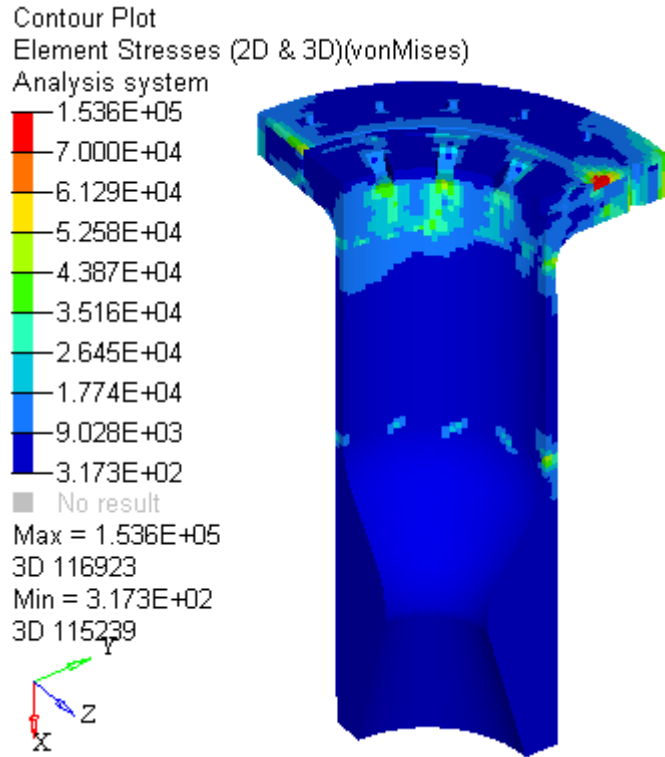
44

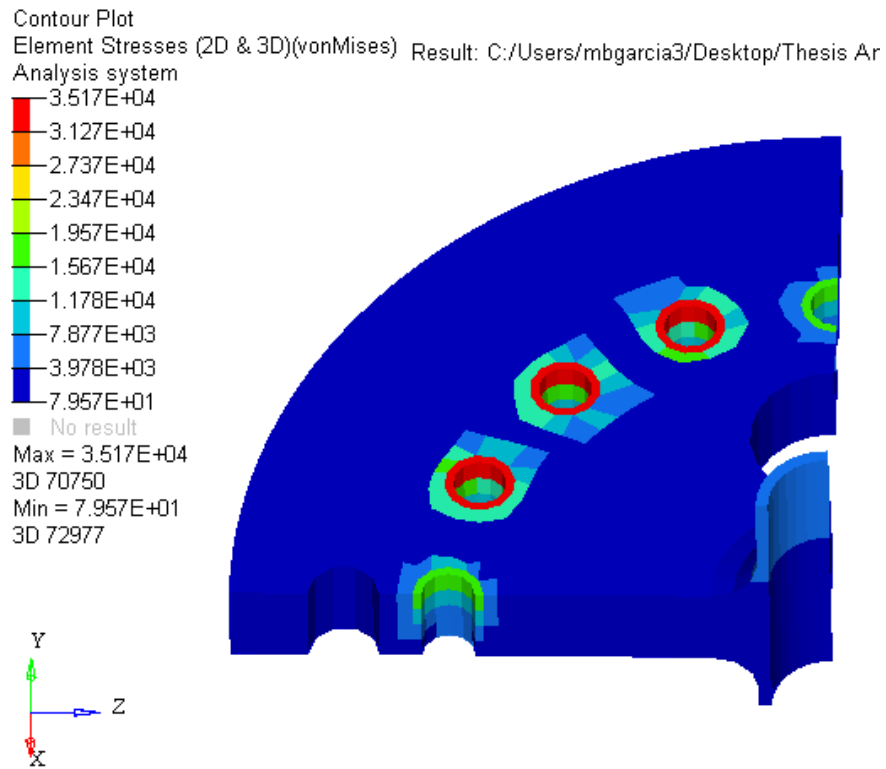Figure 4.20 - Hypermesh chamber combined stresses

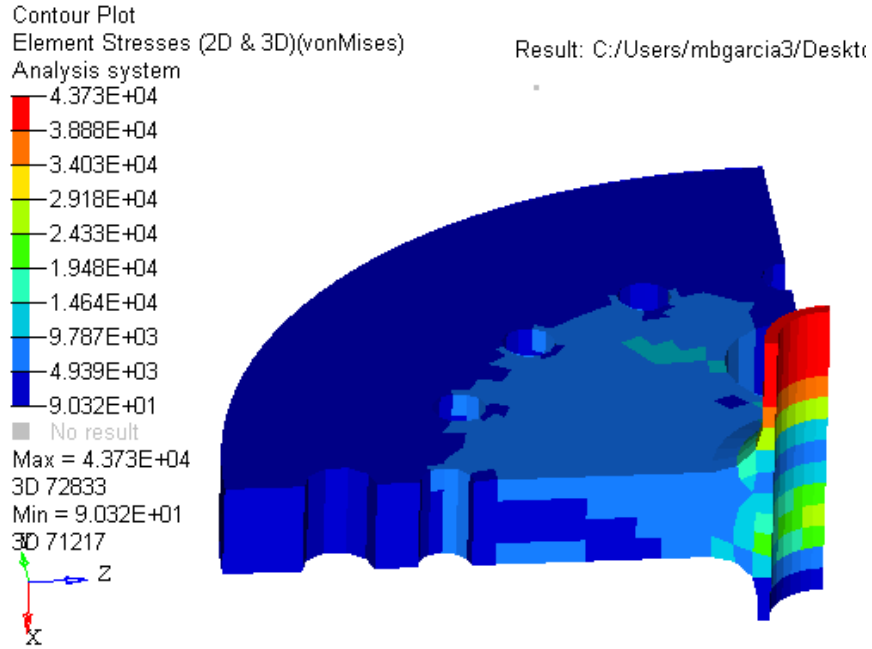Figure 4.21 - Hypermesh pintle manifold mechanical stresses

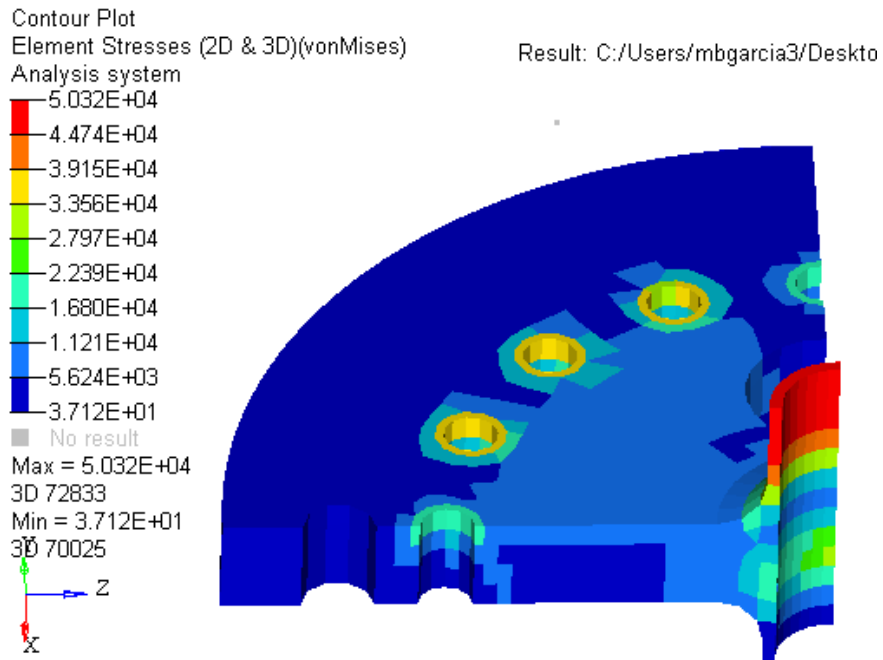Figure 4.22 - Hypermesh pintle manifold thermal stresses

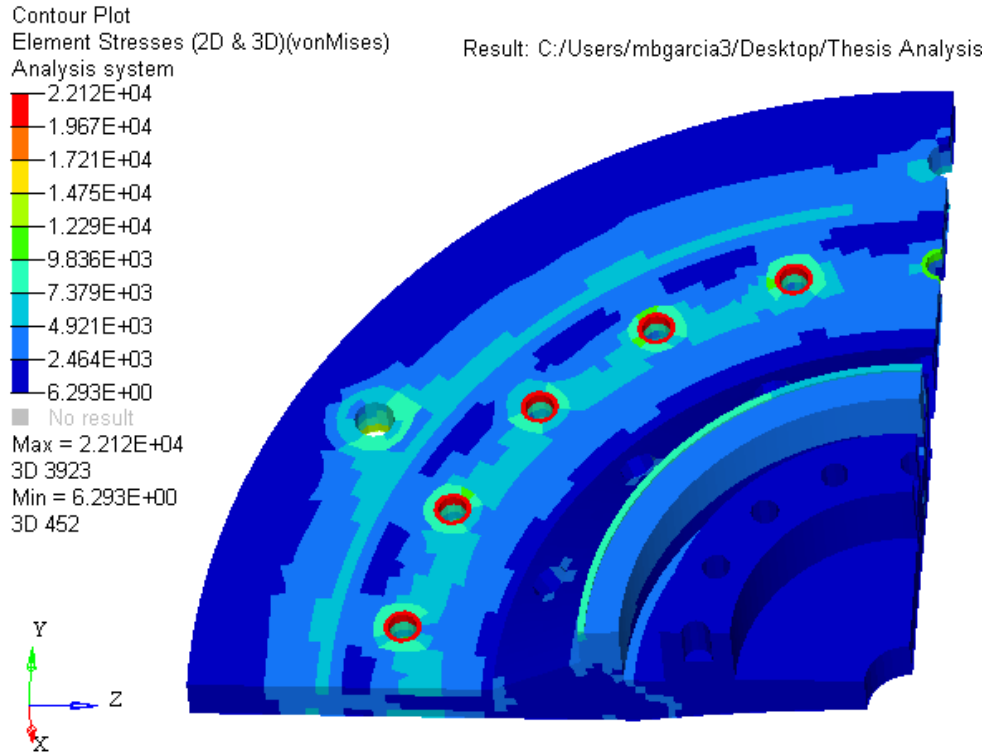Figure 4.23 - Hypermesh pintle manifold combined stresses

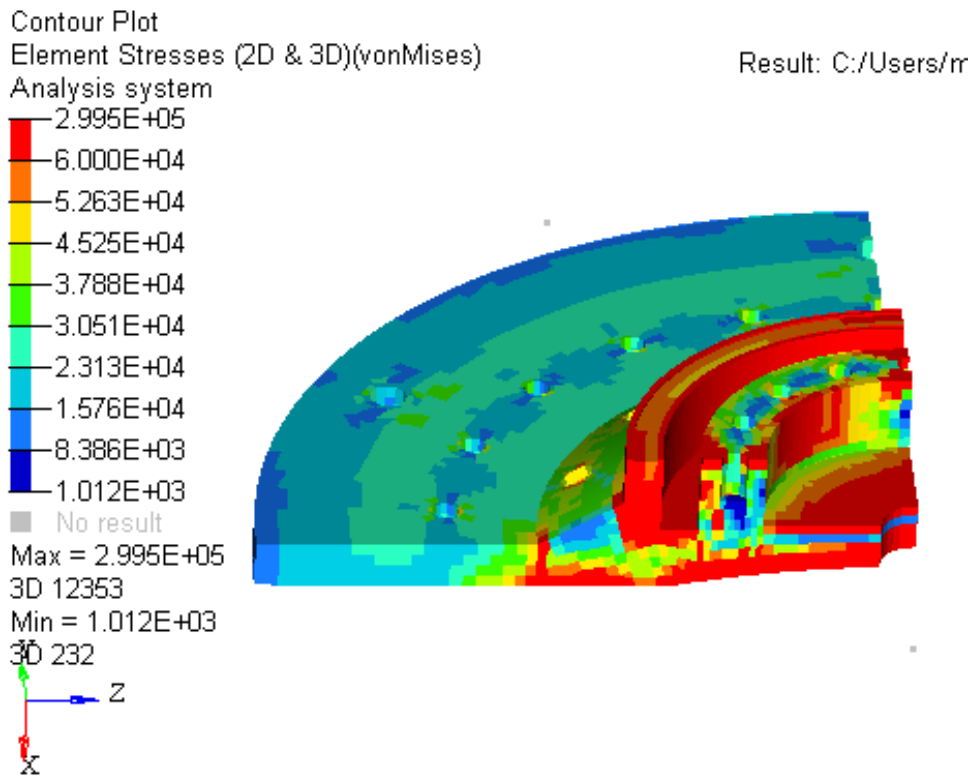Figure 4.24 - Hypermesh injector body mechanical stresses

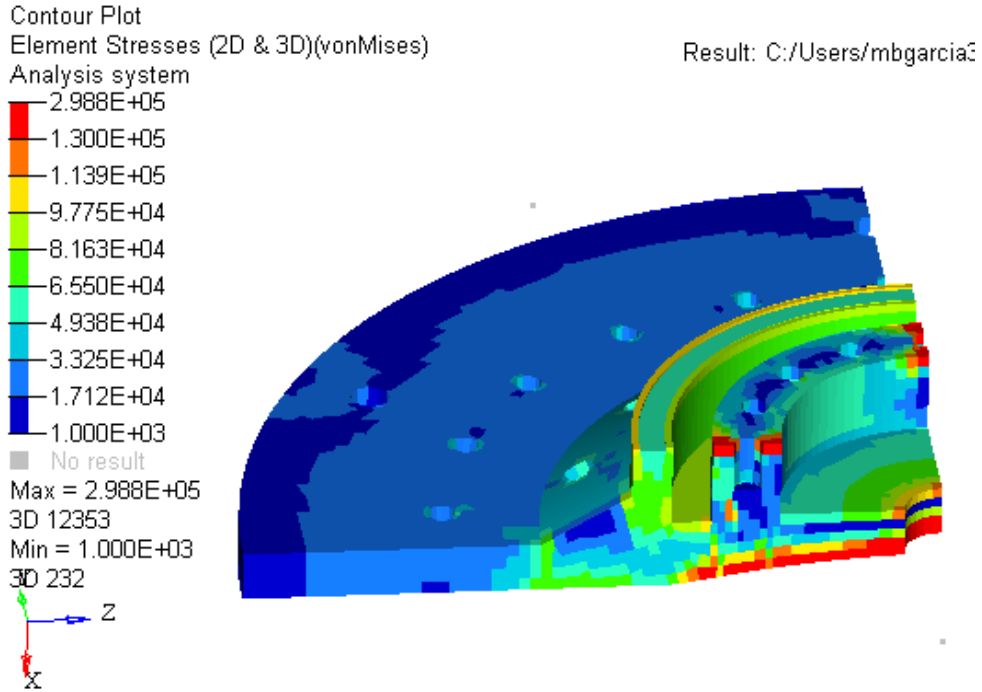Figure 4.25 - Hypermesh injector body thermal stresses

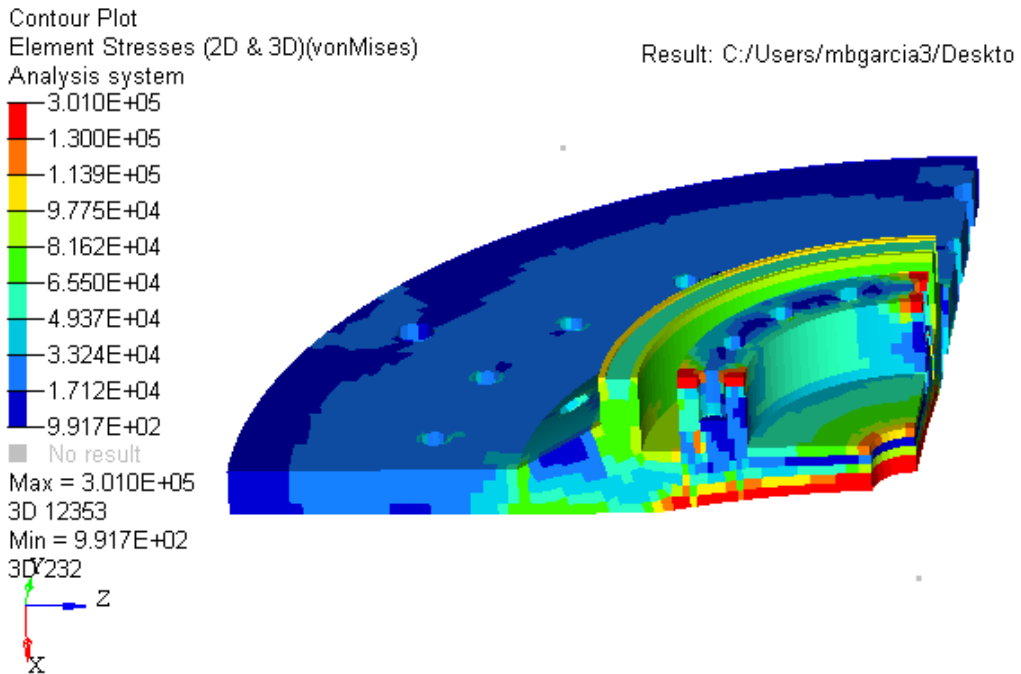Figure 4.26 - Hypermesh injector body combined stresses using Inconel 625

Figure 4.27 - Hypermesh injector body combined stresses using Inconel 718

Table 4.8 - Safety factors of the CROME-X Hypermesh analysis

Component	Max Stress (ksi)	Mechanical Safety Factor	Max Stress (ksi)	Thermal Safety Factor	Max Stress (ksi)	Safety Factor (combined)	
Chamber	22.2	7.0	70	2.2	75	2.1	
Injector	22.1	2.7	300	0.2	130	625: 0.5	718: 1.2
Pintle Manifold	35.2	1.7	43.7	1.37	50.3	1.2	

Based on the results, both the chamber and the pintle manifold meet the required safety factor. The chamber achieved a safety factor of 2.1 for the combined loading case and the pintle manifold achieved a safety factor of 1.2. The injector body was the only component to not meet the required safety factor. However, there is a "fix" that can be done so that the injector body could meet the safety factor. Currently, the selected material for the injector body is Inconel 625. This material at 1000°F has a yield strength of 60 ksi. The selected material for the chamber is Inconel 718 which has a yield strength of 155 ksi at the same temperature. If the injector body material was changed to Inconel 718, then the component could achieve the desired safety factor.

Chapter 5: Mechanical and Thermal Analysis - Abaqus

5.1 APPROACH

The mesh size that was used for the Abaqus model was 0.125. The mesh process for an Abaqus model is the same process has described in Chapter 4. The most crucial difference is when loading Hypermesh, select Abaqus option. If Abaqus is selected after the meshing process begins, Abaqus CAE may not accept the elements when the .inp file from Hypermesh is imported. For the Abaqus model, contact interactions were created. Creating the contact interactions is the same process as described in section 2.4. Once the mesh, properties, and contacts are created, the model can be exported as an .inp file to input into Abaqus CAE. Be sure not to use spaces, capital letters, or periods in the name because this could disrupt the import process. The symmetry boundary conditions, mechanical loads, and thermal loads that are applied in the model can be seen in the tables below. The explanation on creating these loads are explained in later sections.

Table 5.1 – Abaqus chamber mechanical boundary conditions
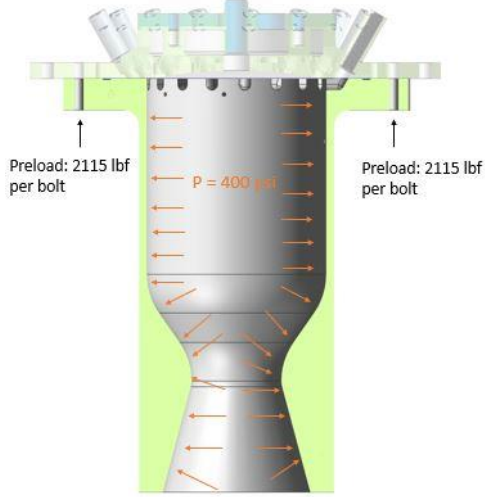
Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
 • SPC's
     o Y-sym: 2,4,6
     o Z-sym: 3,4,5
     o X-direction: 1
	• Chamber Pressure
	 • Chamber Bolt
	     o $A = \frac{\pi}{4}(\frac{7}{16})^2$ (based on bolt head diameter)
	     o $P = \frac{F\,(Preload)}{A}$

Table 5.2 - Abaqus chamber thermal boundary conditions

Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
    ○ Y-sym: U2 = R1 = R3 = 0
    ○ Z-sym: U3 = R1 = R2 = 0
    ○ X-direction: U1 = 0
	• Chamber Temperature
	• Acoustic Cavity Temperature
	    ○ 80% of chamber temperature
	• Convection to ambient

Table 5.3 - Abaqus injector mechanical boundary conditions
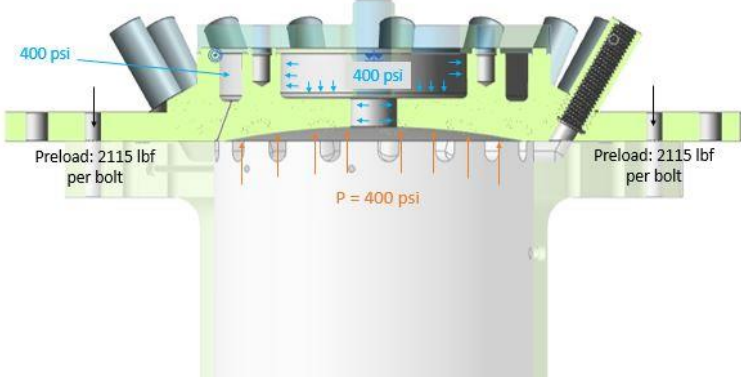
Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
    ○ Y-sym: 2,4,6
    ○ Z-sym: 3,4,5
    ○ X-direction: 1
	• Chamber Pressure
	• Propellant Pressure
	• Chamber Bolt
	    ○ $A = \frac{\pi}{4}\left(\frac{7}{16}\right)^2$ (based on bolt head diameter)
	    ○ $P = \frac{F\,(Preload)}{A}$

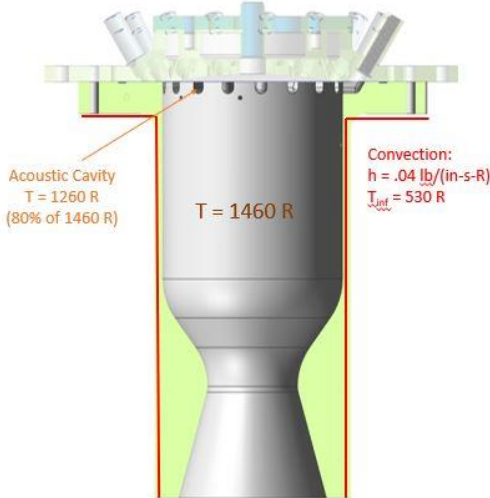Table 5.4 - Abaqus injector thermal boundary conditions

Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
   o Y-sym: U2 = R1 = R3 = 0
   o Z-sym: U3 = R1 = R2 = 0
   o X-direction: U1 = 0
	• Fuel Temperature
	• Oxidizer Temperature
	• Acoustic Cavity Temperature
	   o 80% of chamber temperature
	• Convection from chamber temperature
	• Convection to ambient

Table 5.5 - Abaqus pintle manifold mechanical boundary conditions
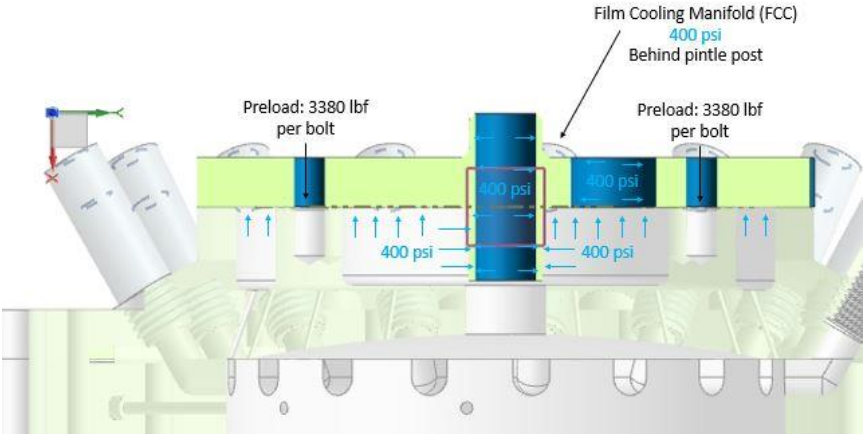
Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
   o Y-sym: 2,4,6
   o Z-sym: 3,4,5
   o X-direction: 1
	• Propellant Pressures
	• Injector Bolt
	   o $A = \frac{\pi}{4}\left(\frac{7}{16}\right)^2$ (based on bolt head diameter)
	   o $P = \frac{F\,(Preload)}{A}$

Table 5.6 - Abaqus pintle manifold thermal boundary conditions
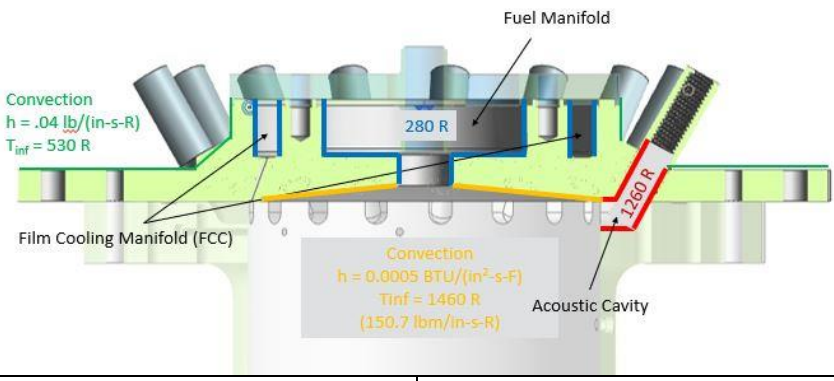
Film Cooling Manifold (FCC)
280 R
Behind pintle post

Oxidizer Port

Convection
h = .04 lb/(in-s-R)
T_{inf} = 530 R

280 R

210 R

Fuel Port

Single Point Constraints (SPCs)	Loads

• Quarter Symmetry
• SPC's
    ○ Y-sym: U2 = R1 = R3 = 0
    ○ Z-sym: U3 = R1 = R2 = 0
    ○ X-direction: U1 = 0
	• Fuel Temperature
	• Oxidizer Temperature
	• Convection to ambient

5.2 BOUNDARY CONDITIONS

The boundary conditions that are used in Abaqus are symmetry constraints, vertical direction constraints, thermal loads, and mechanical loads. The symmetry constraints are necessary because the model is using a quarter cut of the entire geometry. As mentioned before, using a half, quarter, etc. cut of a model reduces time of computation. When originally cutting the geometry in the CAD software, try cutting the sections along the x-, y-, or z- axis. This helps when applying these symmetry boundary conditions. In this quarter symmetry model, two symmetry boundary conditions are used. In Abaqus, the symmetry boundary condition can be applied by right clicking on BC > Create > Symmetry/Antisymmetry/Encastre on the Initial Step. Then change the Select regions for boundary condition to 'by angle' and select the surfaces to be constrained. This process is the same as the surface selecting for symmetry boundary conditions that is briefly described in 4.2 Boundary Conditions. When all the surfaces are selected, click

Done and various options appear in a new window. If the x-axis is perpendicular to the surfaces, select XSYMM, if the y-axis is the perpendicular axis, select YSYMM, and the same goes for the z-axis. An axial constraint must also be applied to the components, in this model, the constraint will be applied in the x-direction. One node on every component that lies within the bolt circle must be constrained in the x-direction or the 1 direction. This is to prevent the model from "flying off" and cause singularities during analysis and it also does not overly constraint it. If a singularity error occurs during the run, the message file will give the ID of a node that is causing the singularity. To resolve this, find the node and constrain it in the axial direction. This error usually occurs to a node that is on a bolt or a screw. [4] To find the node, note the node ID on the error message. Then within Abaqus, select Tool > Display Group > Create > Nodes > Method: Node Labels. Then input the node ID into the Node Label section and check mark the Highlight items in viewport so the node can be highlighted. The node may not be on the outer surface of the component, so the view cut may need to be used to find the node. This action can be accessed by clicking the View Cut Manager icon which is located under the Enable Selection arrow icon. Once the view has been adjusted enough the node has been zoomed into, a boundary constraint can now be made. This time, instead of the Symmetry step, select the Displacement/Rotation step, change the selection region from "by angle" to "individually" and select the zoomed in on node. The displacement constraint should now be made. These boundary conditions are applied both in the mechanical analysis and the mechanical analysis.

5.2.1 Mechanical

For the mechanical analysis, bold preloads and pressures are the loads that are applied, and the element type is 3D stress elements, which is the default setting. There are multiple steps to create a bolt preload on Abaqus. The first step is to create an axis line along the center line of the bolt. This can be done by going to Tools > Datum > Axis, 3 points on circle. Select three

points that run along the edge of the bolt head and a axis should then be created. Repeat this process for every bolt and screw. Right click on Load and select Create. Rename the load and check that the selected step is "Mechanical" and select Bolt Load for the Type for Selected Step. Then, change the selection method to By Angle and select the bolt midsurface that lies between the two components that the bolt is connecting as seen in Figure 5.1 and press okay. The color inside of the area should either be brown or purple. Depending on which color it is, select that color in the next section. In the next section, select the axis that runs along the midline of the bolt that was created earlier. The positive value of the preload can be inputted into the Magnitude box. If it is only half a bolt, input half of the full preload value. The preload arrow directions should be pointing in opposite directions if it was applied correctly. The method and the amplitude remain the default setting. This process will be repeated for every bolt, unfortunately multiple areas not axis can be selected at once. It also may take some time to select the correct area, so patience is critical.
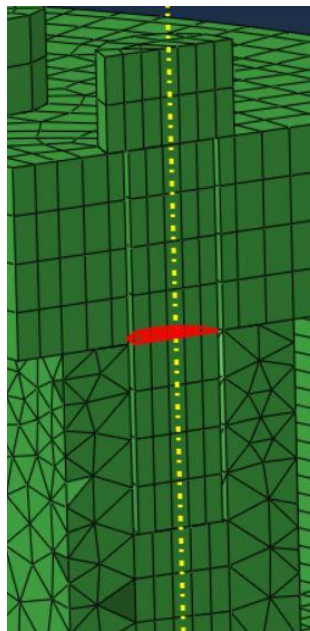

Figure 5.1 - Applying bolt preload in Abaqus

After each preload is created, submit the model for analysis. If the model runs, the loads and boundary conditions are functioning, if there is an error, the cause can be kept track off if one new load is added each time. One error that occurs with preloads is the singularity error.

55

This can be resolved by constraining the node that is causing the singularity as explained earlier in this paper.

The next load type that needs to be created are the pressure loads that are seen in the chamber and in the injector caused by the propellants. Creating the pressure loads is similar to the preloads, start with right clicking on Loads and select create. Use the mechanical load step and change the type to pressure. Then, select the surfaces on which this pressure is to be applied and select multiple surfaces by holding the shift button. The value of the pressure can then be inputted in the magnitude box. The direction that the pressure is applied to should be towards the surface, not away. The pressures can introduce error in the analysis as well, so introduce pressures after all preloads are working and allowing the model analysis to run.

5.2.2 Thermal

When creating the step for a heat transfer analysis, the mechanical step must be suppressed. The two cannot operate at the same time with the current element type selection in Abaqus. The element type for the thermal analysis must be changed to Heat Transfer elements. This can be done by first changing the Module: to Part and then changing it to Mesh. Click Mesh at the top of the tool bar and select Element Type. The selection region can be "individually" and select all the element on the screen by drawing a window around the engine assembly. The elements should turn red when the elements are selected. In the following window, the element type should be changed to Heat Transfer for this analysis. Now that the elements have been changed, the thermal step can be created. Right click the Step and change the Type to Heat Transfer. The Type of Response in the next window needs to be changed to Steady-state. Under the Incrementation tab, input the following as shown in Figure 5.2.
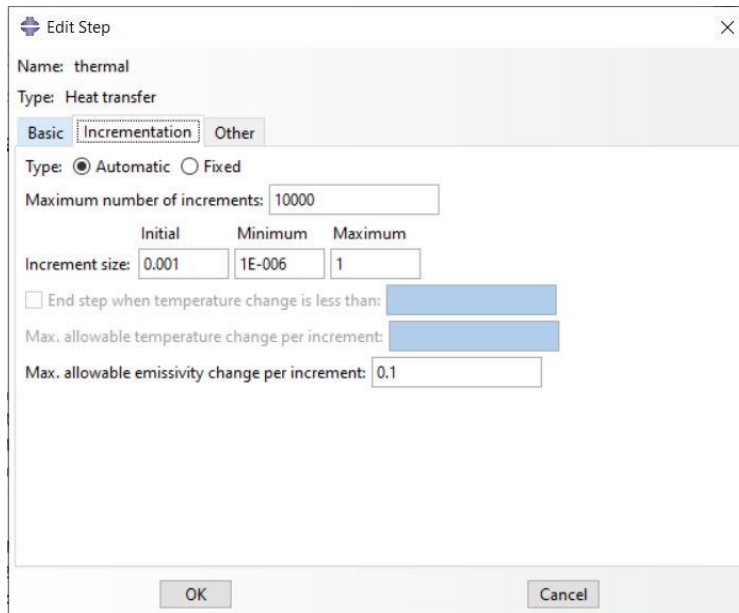
Figure 5.2 - Thermal step for analysis

The two thermal loads that are in the thermal analysis are surface temperatures and convection from component surfaces. To create the surface temperature boundary condition, right flick on BCs, and click Create. Ensure that the Step: is Thermal (or whatever name the thermal step was given) and have Temperature selected in Types for Selected Step. Then select the surfaces on which this particular temperature is to be applied and once all surfaces are chosen press done. The value of the temperature boundary condition is inputted in the Magnitude box and all other sections are left as default. The units of the temperature need to be in Rankine. This process is repeated for every different temperature boundary condition. The convection boundary conditions are created via right clicking Interactions and selecting Create. Change the Type for Selected Step to Surface Film Condition and the Step: should be the thermal step that was created. The heat transfer coefficient is inputted into the Film Coefficient box and the environment temperature (chamber temperature or ambient temperature) is inputted into the Sink Temperature box. An example can be seen in Figure 5.3.

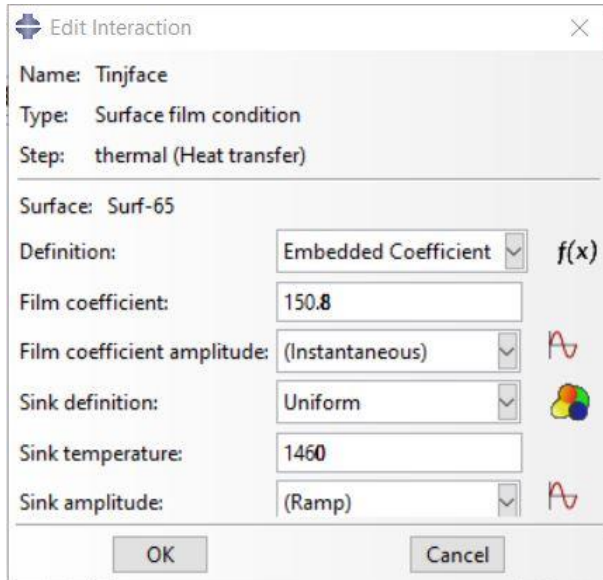Figure 5.3 - Creating a convection interaction for the injector face

The heat transfer coefficient that was applied to the injector face that faces the chamber was provided by experimental data by NASA Johnson Space Center Engineers who were collaborating with Purdue. The experiment is similar to the research that cSETR is conducting so the heat transfer value was translated to use in this analysis [1].

After the temperatures and convection interactions have been created, the job to run the thermal analysis can be created. Right click on Job and select Create. All the options can be left as default. The inputs that are in the job should be the surface temperature boundary conditions, the convection interactions, the contact interactions, and the symmetry boundary conditions. Once this is checked, the job can be submitted for analysis.

5.3 RESULTS

The temperature gradient can be seen in Figure 5.4. The temperature gradient created in Abaqus is comparable to the Hypermesh model, with the change in temperatures being within 10° of each other. When comparing the thermal stress analysis results from Abaqus to Hypermesh, the results were different by an order of magnitude. After manipulating the coefficient of thermal expansion (CTE) in Abaqus, the maximum stress value of the model did

not vary. This led to the conclusion that the model set up was incorrect since manipulating the

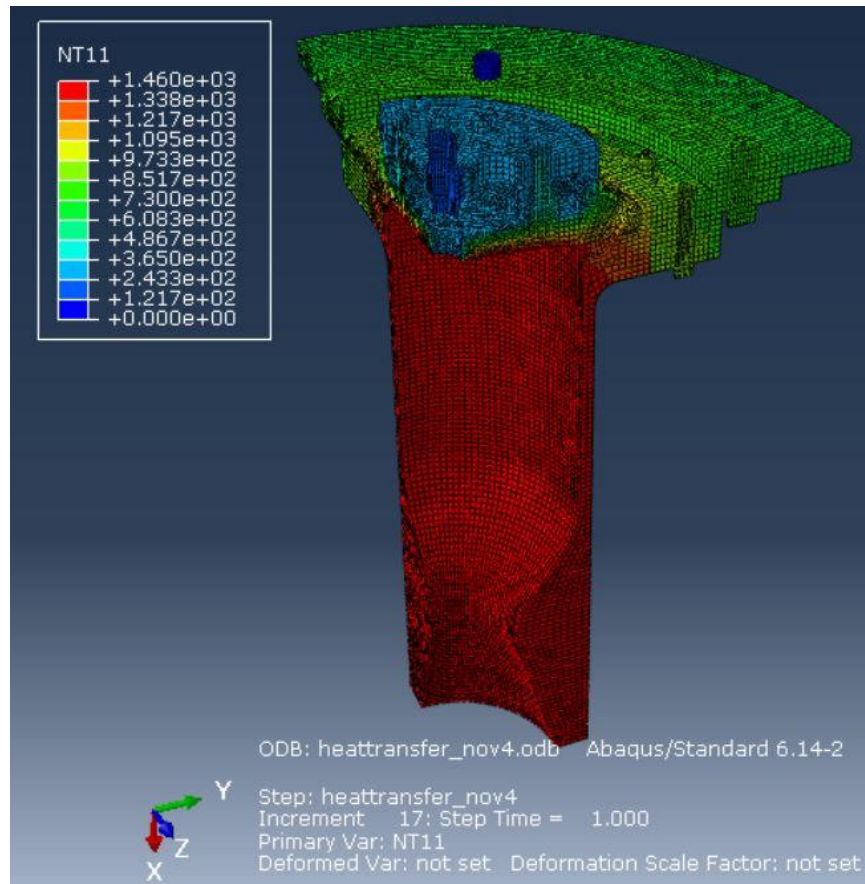CTE should have varied the generated thermal stresses.

Figure 5.4 - CROME-X Abaqus thermal gradient

Chapter 6: Conclusion

6.1 ANALYSIS COMPARISON

For the thermal analysis comparison, the results in Hypermesh outputted a higher max thermal stress compared to the Abaqus analysis. The Hypermesh has a max thermal stress in the injector body of 130 ksi and the max stress which also occurred in the injector body in the Abaqus analysis was 19.3 ksi. This significant difference could be because the Abaqus model models the entire thrust chamber assembly whereas in the Hypermesh analysis, the components are analyzed individually. In this case, the Abaqus model is more reliable model because it is a model of the thrust chamber assembly which is how the components will interact in real life testing. Therefore, based on the Abaqus analysis, all the components in the CROME-X assembly pass and achieve the required factor of safety, 1.2.

6.2 FUTURE WORK

The mechanical analysis for the CROME-X in Abaqus is currently still being trouble-shot so the model can run. There were obstacles that got in the way of completing the analysis which include a delay in obtaining access to the UTEP HPC. The analysis input files are too large for a laptop to handle so these jobs are submitted to the UTEP HPC. Once results are obtained from the mechanical analysis, this thesis will be updated.

Other future work includes completing the manufacturing drawings for the combustion chamber, the injector body, and the pintle manifold. These drawings must be completed and then submitted for manufacturing quotes. The goal is to test the CROME-X engine this Summer 2020, so the drawings need to be completed in January and submitted no later than the first week of February 2020. The drawings are currently being worked on, they just need to be completed and approved.

References

[1] Bedard, Michael J, 2012. "Student Design/Build/Test of a Throttleable LOX-LCH4 Thrust Chamber." In AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, *Purdue University.*

[2] Budynas, Richard Gordon., and J. Keith. Nisbett. 2011. Shigley's Mechanical Engineering Design. 9th ed. Boston: McGraw-Hill Higher education.

[3] Candelaria, Jonathan (2017). Theoretical Acoustic Absorber Design Approach for LOX/LCH4 Pintle Injector Rocket Engines. *The University of Texas at El Paso*.

[4] "Contact." Abaqus Version 6.5 Documentation. Accessed November 7, 2019. https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.5/books/gss/default.htm?startat=ch11s04.html.

[5] Harrje, David T, and Reardon, Frederick H. 1972. "Liquid Propellant Rocket Combustion Instability." In JANNAF, 1972.

[6] "INCONEL alloy 625" *Special Metals Corporation*. August 13, 2013. https://www.specialmetals.com/assets/smc/documents/alloys/inconel/inconel-alloy-625.pdf

[7] "INCONEL alloy 718" *Special Metals Corporation*. September 7, 2007. https://www.specialmetals.com/assets/smc/documents/alloys/inconel/inconel-alloy-718.pdf

[8] Lopez, I. (2017). Design of a 2000 lbf LOX/LCH4 Throttleable Rocket Engine for a Veritcal Lander. *The University of Texas at El Paso*.

[9] Santana Jr., A. 2007. "Acoustic Cavities Design Procedures." *Thermal Engineering* 6, No. 2 (December 2007): 27-33.

[10] Stoddard, F. J. 1993. "Inherent Stability of Central Element Coaxial Liquid-liquid Injectors." *NASA Propulsion Engineering Research Center* 2, (November 1, 1993): 75-76.

Vita

Marissa B. Garcia was born and raised in El Paso, Texas. After graduating from Franklin High school, she accepted the Presidential Excellence academic scholarship to attend the University of Texas El Paso to pursue a Bachelor of Science in Mechanical Engineering. As a rising senior, she accepted an ATLAS intern position at the Johns Hopkins University Applied Physics Lab where she worked in the facilities department. After this internship, she accepted a position as an undergraduate research assistant for the Center of Space Exploration and Technology Research (cSETR) working on the liquid oxygen/ liquid methane propulsion team, She then completed her Bachelor of Science in Mechanical Engineering in May of 2017 where she was awarded the honor of Outstanding Senior in Mechanical Engineering.. In the pursuit of furthering her education, she enrolled in graduate school at the University of Texas at El Paso to pursue a Master of Science in Mechanical Engineering. There, she continued her research in liquid oxygen/liquid methane propulsion systems and during her graduate studies, she accepted multiple internship and co-op positions. One position was with the NASA Johnson Space Center propulsion branch where she assisted in creating system requirements and designing an RCE conditioning system. The second position was a NASA Pathways intern position with the NASA Marshall Space Flight Center. In this program, she rotated into three different branches: the Propulsion Fluid Dynamics branch, the Structural Design, Development, and Analysis branch, and the Propulsion Engine Component Development branch where she worked on a hybrid motor model, an avionics unit bracket design, and helping to lead a 5000 lbf engine test program. Marissa has accepted a job offer with the NASA Marshall Space Flight Center to work as a propulsion engineer after graduation.

Contact Information: Marissa.b.garcia@nasa.gov