DESIGN AND ANALYSIS OF A REUSABLE N₂O-COOLED AEROSPIKE NOZZLE
FOR LABSCALE HYBRID ROCKET MOTOR TESTING

A Thesis
presented to
the Faculty of California Polytechnic State University,
San Luis Obispo

In Partial Fulfillment
of the Requirements for the Degree
Master of Science in Mechanical Engineering

by
Daniel Joseph Grieb
February 2012

© 2012

Daniel Joseph Grieb

ALL RIGHTS RESERVED

COMMITTEE MEMBERSHIP

TITLE: Design and Analysis of a Reusable N₂O-Cooled
Aerospike Nozzle for Labscale Hybrid Rocket Motor
Testing

AUTHOR: Daniel Joseph Grieb

DATE SUBMITTED: February 2012

COMMITTEE CHAIR: Joseph D. Mello, Professor

COMMITTEE MEMBER: William R. Murray, Professor

COMMITTEE MEMBER: Patrick Lemieux, Associate Professor

ABSTRACT

Design and Analysis of a Reusable N₂O-Cooled Aerospike Nozzle for Labscale Hybrid Rocket Motor Testing

Daniel Joseph Grieb

A reusable oxidizer-cooled annular aerospike nozzle was designed for testing on a labscale PMMA-N₂O* hybrid rocket motor at Cal Poly-SLO.[†] The detailed design was based on the results of previous research involving cold-flow testing of annular aerospike nozzles and hot-flow testing of oxidizer-cooled converging-diverging nozzles. In the design, nitrous oxide is routed to the aerospike through a tube that runs up the middle of the combustion chamber. The solid fuel is arranged in an annular configuration, with a solid cylinder of fuel in the center of the combustion chamber and a hollow cylinder of fuel lining the circumference of the combustion chamber. The center fuel grain insulates the coolant from the heat of the combustion chamber. The two-phase mixture of nitrous oxide then is routed through channels that cool the copper surface of the aerospike. The outer copper shell is brazed to a stainless steel core that provides structural rigidity. The gaseous N₂O flows from the end of aerospike to provide base bleed, compensating for the necessary truncation of the spike. Sequential and fully-coupled thermal-mechanical finite element models developed in Abaqus CAE were used to analyze the design of the cooled aerospike. The stress and temperature distributions in the aerospike were predicted for a 10-sec burn time of the hybrid rocket motor.

Keywords: Aerospike Nozzle, Finite Element Analysis, Hybrid Rocket, Two-Phase Flow, Nitrous Oxide

* PMMA stands for polymethyl methacrylate, a thermoplastic commonly known by the brand name Plexiglas®. N₂O is the molecular formula for nitrous oxide.

[†] California Polytechnic State University, San Luis Obispo

ACKNOWLEDGMENTS

I would like to acknowledge the hard work of everyone on the hybrid rocket research team in the Mechanical Engineering Department at Cal Poly. Dr. William Murray has provided excellent leadership and worked to maintain a steady stream of funding. Dr. Joseph Mello has offered his expertise in finite element analysis and rocket design. Dr. Patrick Lemieux has provided valuable guidance in the fields of gas dynamics and compressible flow. Terry Cooke's work on two-phase rocket nozzle cooling and his management of the rocket motor testing have been indispensable. Jim Gerhardt has been the excellent machinist that makes building our rocket motors possible. Dr. Mike Kerho of Rolling Hills Research Corporation provided valuable support in the area of computational fluid dynamics. I would also like to thank NASA Dryden Flight Research Center, and our technical monitor, Nalin Ratnayake, for the continued financial support that has made this research possible.

TABLE OF CONTENTS

	Page
LIST OF TABLES	ix
LIST OF FIGURES	x
1. Background	1
<i>Aerospike Nozzles.....</i>	<i>2</i>
<i>Hybrid Rockets</i>	<i>2</i>
<i>N₂O Cooling of Rocket Nozzles</i>	<i>4</i>
2. Development of a Preliminary Design Concept	6
3. Aerospike Design Requirements	15
<i>Hybrid Rocket Combustion Design Conditions</i>	<i>15</i>
<i>Aerospike Contour Design</i>	<i>16</i>
<i>Design Heat Loads</i>	<i>16</i>
4. Idealized N₂O Coolant Process Path	20
5. Preliminary Design Stress Calculations	23
<i>Copper Shell Thickness</i>	<i>23</i>
<i>Channel Depth.....</i>	<i>25</i>
6. Finalized Concept Design.....	27

7. Detailed Coolant Flow Analysis.....	31
<i>The Coolant Supply State</i>	<i>31</i>
<i>The Supply Lines</i>	<i>31</i>
<i>The Inlet Pipe</i>	<i>32</i>
<i>The Coolant Channels.....</i>	<i>35</i>
The Long Coolant Channels	36
The Medium and Short Coolant Channels	37
<i>The Outlet Channels.....</i>	<i>38</i>
<i>The Coolant Process Path.....</i>	<i>39</i>
8. Strategy for Finite Element Modeling of the Aerospike	43
<i>Insulator Analysis.....</i>	<i>43</i>
<i>Throat Region Analysis</i>	<i>44</i>
9. Finite Element Analysis Results	46
<i>Insulator Analysis.....</i>	<i>46</i>
The Model.....	47
The Results	52
<i>Throat Analysis</i>	<i>61</i>
The Model.....	61
Results	65
Ways to Improve the Model	80

10. Summary of the Design	83
11. List of References	86
Appendices.....	89
A. Aerospike Cooling Channel Preliminary Analysis	89
<i>Assumptions About the Cooling Model for N_2O.....</i>	<i>89</i>
<i>Analytical Comparison of Coolant Channel Designs</i>	<i>92</i>
B. Development of Fully Coupled Thermal/Mechanical FEA	97
<i>Theory of Thermomechanical Coupling.....</i>	<i>97</i>
<i>Analysis of an Insulator in a Rocket Nozzle.....</i>	<i>102</i>
Modeling the Insulator.....	103
Results	105
Conclusions	111

LIST OF TABLES

	Page
Table 1: A summary of the states of the coolant at the inlet and outlet of each region of the flow path.	42
Table 2: The loads applied to the insulator model.	51
Table 3: The loads used in the thermal model of the throat region. Note that $z=0$ at the throat.	63
Table 4: The loads applied to the mechanical model of the throat region.	64

LIST OF FIGURES

	Page
Figure 1: An early Cal Poly concept of a hybrid rocket motor with an aerospike nozzle. Courtesy of Jim Gerhardt.	1
Figure 2: The geometry of the aerospike used for cold flow tests at Cal Poly.	6
Figure 3: A comparison of the streamlines and mach contours of two aerospike geometries. Aerospike 5 on the top has the geometry used in the cold flow testing, while Aerospike 6 has modifications to smooth the converging region.	7
Figure 4: Local heat transfer coefficients, h , determined from CFD. The plot on the left shows the heat transfer into the spike while the plot on the right shows the heat transfer into the graphite liner on the outer wall of the nozzle.	8
Figure 5: Streamlines and Mach contours for three instances in time throughout a standard burn. The prominence of the recirculation regions in the wake of the fuel grains can be seen in the initial configuration. These eddies are reduced throughout the burn.	9
Figure 6: Local heat transfer coefficients, h , determined from CFD for three instances during the course of a burn. The plot on the left shows the heat transfer into the spike while the plot on the right shows the heat transfer into the graphite liner on the outer wall of the nozzle.	10
Figure 7: The final concept for the geometry of the aerospike. The green region represents the net shape of the spike assembly. The yellow regions represent the end of inner and outer fuel grains for an annular combustion chamber.	12
Figure 8: The conceptual design of the spike assembly. The stainless steel core is shown in gray. The insulator is shown in black. The outer sheath will be made from copper.	13
Figure 9: The aerospike contour calculated for $NPR=20$ and $\gamma=1.14$ at $P_{atm}=14.696$ psia. The blue line represents the contour of the spike, while the red line shows the position of the throat of the nozzle.	16
Figure 10: The heat transfer coefficient, h , on the surface of the aerospike as computed by the FLUENT model. The throat of the nozzle is located at $x=0$	17
Figure 11: The heat load being transferred into the aerospike per unit length, calculated based on a constant surface temperature of $400\text{ }^{\circ}\text{F}$. This represents the amount of heat that must be transferred into the coolant at each aerospike cross-section.	18

Figure 12: The idealized process path of the N ₂ O coolant flowing through the aerospike nozzle. The cooling capacity Δh represents the maximum amount of heat the coolant is capable of absorbing without the quality rising about the chosen design limit of $x=0.5$	22
Figure 13: The load on the copper shell due to coolant pressure can be modeled as a beam of infinite width (into page) under uniform distributed loading.	24
Figure 14: The sketch of the top view of a wide beam. For the case of long coolant channels, it is assumed that the ratio of width to span is $a/b=\infty$	24
Figure 15: The final concept of the aerospike assembly, consisting of a stainless steel core covered with a copper shell and protected by a graphite or ceramic insulator upstream.	27
Figure 16: The stainless steel core has 20 grooves cut in it with a 5/64 inch ball end mill. Those grooves act a coolant channels. There are three different lengths of grooves: 5 long grooves, 5 medium grooves, and 10 short grooves.	28
Figure 17: The long channel can be seen along the upper edge of the aerospike assembly, just beneath the copper shell. The medium channel can be seen running along the bottom edge of the aerospike assembly. An injector can be seen upstream in the flow, at the entrance to the stainless steel core. The injector and the inlet and exit ports to the channels will be sized to obtain the correct coolant mass flow rate through the aerospike.	29
Figure 18: The short coolant channels are used to cool only throat region, where the heat load is highest. The radial channels routing the cooling to and from the outer grooves will be sized to balance the mass flow between the various channel lengths.	30
Figure 19: The plot shows the calculated head loss and dynamics pressures at the exit of the inlet pipe for various exit pressures. The solution is found at the intersection of the pressure drop, dP , and the total losses, $HL+Gdu$. This occurs at 458 psia.	34
Figure 20: The representation of the calculated process paths for the three different lengths of coolant channels.	40
Figure 21: The detailed calculated process path for flow through the short channels.....	40
Figure 22: The detailed calculated process path for flow through the medium length channels.	41
Figure 23: The detailed calculated process path for flow through the long channels.....	41
Figure 24: The region shown was analyzed using a fully coupled 2D axisymmetric technique in order to determine the best material to be used as an insulator.	44
Figure 25: The 3D sector analysis of the throat section includes the curved surface of the aerospike up to the plane where the spike is shown truncated above, at $x=0.06$ inch.	45

Figure 26: The assembled parts modeled for the analysis of the insulator region. The parts are partitioned to improve the mesh quality. There are four separate parts: the spike core, the insulator, the support rod, and the center fuel grain.....	48
Figure 27: The thermal loads applied to the insulator region.	50
Figure 28: The mechanical loads and boundary conditions that were applied to the insulator region.	51
Figure 29: The meshed assembly of the insulator region.	52
Figure 30: Deformed contour plot of the stress distribution in the insulator region of the aerospike after a 10 sec burn.	53
Figure 31: Deformed contour plot of the temperature distribution in the insulator region of the aerospike after a 10 second burn. The four nodes highlighted show the locations from which temperature histories were taken.....	54
Figure 32: The history plot of nodal temperatures at four important locations in the insulator region. Notice that the temperature on the surface of the insulator does not reach the maximum operating temperature of 3090°F for AD-96 CoorsTek Alumina.	55
Figure 33: The maximum principal stress in the insulator at t=0.9 sec into the burn. The colored contours represent maximum principal stresses that exceed the desired peak stress of 26750 psi, which corresponds to a safety factor of 1.2. The peak stress exceeds the desired limit of 26750 psi, but does not reach the ultimate strength of 32100 psi. The highlighted node on the inner surface was used as a location for the history plot.	57
Figure 34: The history plot above shows the maximum principal stress on the inner surface of the insulator at the node highlighted in Figure 33. The peak maximum principal stress occurs at approximately 0.9 sec into the burn when stress reaches 30092 psi, or 93.7% of the ultimate tensile strength.....	58
Figure 35: The maximum principal stress in the insulator at the upstream end after 2.2 seconds of burn. The stresses at the stress concentration exceed the ultimate tensile strength of alumina, which is 32100 psi. This result is the driving reason behind designing the insulator to be made of two separate pieces of alumina.	59
Figure 36: Contour plot of the maximum principal stress in the downstream end of the insulator at t=2.2 seconds. The peak stress exceeds the ultimate tensile strength of the alumina, which inspired the decision to redesign the end to be stepped instead of tapered.....	60
Figure 37: The assembled geometry of the throat region that was analyzed, consisting of a copper shell brazed to a stainless steel core. The parts are partitioned to improve the mesh quality.	62
Figure 38: The loads and boundary conditions used for the thermal analysis of throat region of the aerospike.	63

Figure 39: The mechanical loads and boundary conditions used for the stress analysis of the throat region of the aerospike. These loads are in addition to the mapping of the nodal temperatures that were calculated in the preceding thermal analysis.	64
Figure 40: The meshed throat model. The copper shell is made up of 18176 linear hexahedral and 576 linear wedge elements, while the stainless steel core is made up of 48047 linear tetrahedral elements.....	65
Figure 41: Deformed contour plot of the temperature in the throat region after a 5 second burn.....	67
Figure 42: The history plot above shows the copper surface temperature at the throat as a function of time for three different values of the heat transfer coefficient, h . The units of h are $\text{BTU}/\text{in}^2\text{-s-}^\circ\text{R}$	68
Figure 43: Deformed contour plot of the von Mises stress distribution in a 3D sector analysis of a cooling channel at the throat after a 5 second burn.	69
Figure 44: Deformed contour plot of the von Mises stress in the copper shell after a 5 second burn.....	69
Figure 45: Deformed contour plot of the plastic strain in the copper shell after a 5 second burn.....	70
Figure 46: Deformed contour plot of the plastic strain in the copper shell after a 5 second burn, viewed from the bottom. Notice that the most yielding occurs at the edge of the brazed surface.	71
Figure 47: Deformed contour plot of the von Mises stress in the copper shell, viewed from the bottom side of the shell. There are two nodes of interest highlighted: one in the middle of the braze and one at the edge of the braze.	72
Figure 48: History plot of the von Mises stress at the two nodes of interest in the braze. The steep discontinuity in the stress occurs when the motor is turned on.	73
Figure 49: History plot of the radial stress, S_{22} , at the two nodes of interest in the braze. The S_{22} stress represents the potential cause of “peel” in the brazed joint.....	73
Figure 50: Deformed contour plot of the compressive axial stress, S_{11} , in the stainless steel core. The steel webs are in compression due to the fact that they are much hotter than the inner stainless steel, which pulls the webs into compression.....	75
Figure 51: Deformed contour plot of the radial stress, S_{22} , in the stainless steel core. The cooled surface of the web (on the sides of the channel) has the highest tensile stress, as it is being stretched by the hotter material that makes up the middle of the stainless steel webs. A section is cut away, revealing the poor quality of the internal tetrahedral meshing.....	76

Figure 52: Deformed contour plot of the von Mises stress in the stainless steel core. The steel located at the braze joint is one of the regions of highest stress, as expected. The high stress at the bottom of the channel was not expected and may be a result of the three dimensional nature of the problem. The highlighted node in the middle of the brazed surface atop the web on the left was used to plot a history of the stresses.....	77
Figure 53: History plot of the von Mises stress at the node in the middle of the brazed surface, highlighted in Figure 52, for the three values of nitrous oxide heat transfer coefficient, h , given in $\text{BTU}/\text{in}^2\text{-s-}^\circ\text{R}$. It can be seen that the two cases of higher heat transfer coefficients, the stress in the braze remains high, while in the case of the low heat transfer coefficient the material reaches much higher temperatures and yields much more, resulting in lower stresses.....	78
Figure 54: Contour plot of the compressive stress in the stainless steel core in the S33 direction, which represents hoop stress. The regions in compression in the S33 direction are hot and want to grow circumferentially, but are being restrained by material within the core that is cooler.....	79
Figure 55: Plastic strain in the stainless steel webs.	80
Figure 56: The geometry that could be modeled to take advantage of another plane of symmetry in the analysis. This should cut the number of elements approximately in half.....	81
Figure 57: The parameter plotted above is proportional to the total heat transferred into the nitrous oxide. As can be seen from the plots, a higher number of channels results in more cooling. Also, channels cut with a ball end mill can cool more than square channels of the same size.	94
Figure 58: The plot above shows the dimensionless parameter that is proportional to the cooling efficiency of the coolant channel designs. As can be seen above, square bottomed channels cool more efficiently than channels cut with ball end mills for a given channel width. Also, having more channels results in more efficient cooling.	94
Figure 59: The parameter plotted above is used to maximize the total cooling as well as cooling efficiency. The red arrow points to a channel width of $5/64$ inch, which corresponds with a ball end mill that is readily available for purchase. It is the size that maximizes the parameter without requiring custom tooling.	95
Figure 60: A cross-sectional slice of the spike showing 20 coolant channels cut with a $5/64$ " ball end mill.	96
Figure 61: The geometry of the graphite shell covering the stainless steel cylinder.....	103
Figure 62: The axisymmetric model of the steel cylinder and graphite shell with a mesh size of 0.0625 in.	104

Figure 63: The nodal temperature contours and displacement of the steel and graphite cylinders after 1.5 seconds. The gap begins to shrink after this time step as the steel begins to heat up.....	106
Figure 64: The nodal temperature distribution in the assembly after 10 seconds.	107
Figure 65: The distribution of the hoop stress in the assembly after 10 seconds. Notice that the peak hoop stress occurs where the coolest part of the steel is pulled to a larger diameter by the hotter material outside of it.....	107
Figure 66: Convergence study of the temperature on the outer surface of the graphite after 10 seconds.	108
Figure 67: Convergence study of the hoop stress on the outer surface of the graphite after 10 seconds.	109
Figure 68: Plot of the gap size as a function of time for various mesh sizes for the fully coupled analysis, as well as one mesh size for the sequential analysis. ..	110
Figure 69: The temperature difference at the contact between the steel and the graphite for case 1.	111
Figure 70: The temperature difference at the contact between the steel and the graphite for case 2.	111

1. Background

A research team in the Mechanical Engineering Department at California Polytechnic State University in San Luis Obispo has been working on the development of hybrid rocket technologies. With the continued support provided by funding from NASA Dryden Flight Research Center, the Cal Poly Mechanical Engineering Department has developed an infrastructure for building and testing lab-scale hybrid rocket motors. Past research, performed in conjunction with Rolling Hills Research Corporation in El Segundo, CA, has studied both the flow characteristics of annular aerospike rocket nozzles,¹ and the use of nitrous oxide as a coolant for traditional converging-diverging rocket nozzles.² The culmination of these research efforts has led to the funding of a project to design, build, and test an annular aerospike that is cooled using N_2O .³

Figure 1: An early Cal Poly concept of a hybrid rocket motor with an aerospike nozzle.
Courtesy of Jim Gerhardt.

Aerospike Nozzles

Aerospike rocket nozzles are designed for consistent performance over a wide range of ambient pressures. Traditional converging-diverging nozzles have a single ambient pressure at which the rocket exhaust gases are neither over-expanded nor under-expanded. As the operating conditions move away from the design nozzle pressure ratio (NPR), a shock or an expansion fan will form at the exit plane of the converging-diverging nozzle. These result in reductions in the efficiency of the nozzle. An aerospike nozzle does not have a solid geometry defining the outer limits of the flow path in the supersonic region of the flow. Instead it allows the exhaust gases to expand freely beyond the throat, via the mechanism of a Prandtl-Meyer expansion fan.

As the name suggests, an aerospike nozzle has a spike in the center of the nozzle. From the throat, the innermost streamlines of the flow follow the contours of the spike, gradually being turned in the axial direction. At the design ambient pressure, the outermost streamlines turn sharply in the axial direction immediately as they leave the throat, a result of being suddenly exposed to the ambient pressure. The path of outermost streamlines is dependent on the ambient pressure in which a given aerospike nozzle is operating. However, the innermost streamlines are unaffected by the ambient pressure and always follow the same path along the contour of the aerospike. Due to the flow of the expanding exhaust gases being dependent on ambient pressure, an aerospike nozzle is capable of providing altitude compensation during flight.

Hybrid Rockets

Hybrid rockets use a solid fuel combined with a liquid oxidizer. This allows hybrid rockets to combine advantages of both solid fuel and liquid fuel rockets.

Hybrid rockets began to interest experts at NASA in the mid-1960s. These rockets have many advantages over both solid and liquid rockets. Hybrid rockets are safe to produce, cheap to manufacture, and they are inherently different from their predecessors. This characteristic means that there is a broad spectrum of research to be done on the subject.

Safety is always a concern when working with rockets. Solid rockets are fueled by extremely explosive materials. Once a solid rocket has been lit, it cannot be put out because the oxidizer and fuel are both mixed into the solid material. Intense caution is required when working with these fuels. Liquid rockets can be dangerous because they require the storage of highly flammable liquids at very high pressures. Hybrid rockets, on the other hand, keep the solid fuel separate from the oxidizer until the rocket is actually being fired. This allows the combustion to be controlled by increasing or decreasing the flow of oxidizer to the fuel. The oxidizer being used herein is nitrous oxide, which is safer to store than the liquid oxygen and liquid hydrogen often used in liquid rockets.

Two different solid fuels are being used in the experiments at Cal Poly: Plexiglas[®] and HTPB. HTPB is a synthetic rubber, similar to the material in car tires. Neither material is dangerous. They are easy to store, manufacture, and handle. No special care is required with these materials. This alone, is a major advantage when performing laboratory tests on rockets and can be advantageous for flight tests as well as future commercial use. This has been demonstrated by Scaled Composites' SpaceShipOne, which was powered by a hybrid rocket motor similar to the one being used at Cal Poly.

Money is a constraint when working with expensive materials and equipment. The fuel in solid and liquid rockets tends to be extremely expensive and hard to come by. Few companies deal with such volatile materials. The materials also need to be stored with

caution. Proper temperature and pressure are crucial to prevent the materials from exploding. The cost of building the proper facilities to handle the needs of these fuels is high enough to prevent most universities from participating in any research involving solid or liquid rockets. However, the fuels in use at Cal Poly are cheap and very easy to purchase. HTPB is around \$10 a pound and Plexiglas[®] is around \$40. This is considerably cheaper than the solid and liquid alternatives. Machining these fuels is also very easy. The cheap cost of materials makes this project feasible.

One disadvantage of hybrid rockets is the low specific impulse. The specific impulse of hybrid rockets falls between the high specific impulse of liquid rockets and the lower specific impulse of solid rocket motors.⁴ This means that liquid rockets achieve more impulse, or thrust times burn time, per pound of fuel burned than hybrid or solid motors. However the density of liquid fuels is relatively low, so hybrid rockets can be comparable to liquid rockets in achieving density impulse.⁵ Another disadvantage of hybrid rockets is that there can be a high residual fuel mass fraction because not all of the fuel grain is consumed during the burn.

N₂O Cooling of Rocket Nozzles

Previous research at Cal Poly has investigated the use nitrous oxide as a coolant for hybrid rocket nozzles.² Nitrous oxide has potential as a good coolant because it is readily available on a hybrid rocket when it is used as the oxidizer. It is self-pressurizing and can flow into the coolant passage as a liquid. The heat of vaporization is then used to absorb the heat being transferred from the exhaust gases into the nozzle. As the nitrous oxide flows through the cooling passages it is a two-phase fluid, increasing in quality as more

heat is transferred into the fluid. Through previous research, it has been determined that this process can be modeled as the combination of Rayleigh and Fanno flows.⁶

2. Development of a Preliminary Design Concept

The cooled aerospike design team at Cal Poly has been focusing its efforts on determining an optimum geometry for the hot side flow path upstream of the throat. The goal was to determine if there was some sort of shape that could minimize the amount of cooling capacity that the aerospike would require for survival in a hot fire test. A literature search failed to reveal any useful information for the design of plug or aerospike nozzles in the converging region. After studying the nature of the problem it was determined that there is a trade-off between lowering the local heat transfer coefficients and maintaining a minimal spike surface area.

Figure 2: The geometry of the aerospike used for cold flow tests at Cal Poly.

Starting from the aerospike geometry used in the cold flow testing at Cal Poly (shown in Figure 2), the shape of the converging section was elongated in hopes of achieving

smoother streamlines and a more constant acceleration of the flow. Figure 3 shows the flow fields of both the cold flow geometry (labeled Aerospike 5) and a design with a smoother flow in the converging region (Aerospike 6).

Figure 3: A comparison of the streamlines and mach contours of two aerospike geometries. Aerospike 5 on the top has the geometry used in the cold flow testing, while Aerospike 6 has modifications to smooth the converging region.

The computational fluid dynamic model of the flow fields performed at RHRC confirmed that the smoother streamlines of Aerospike 6 lead to reduced local heat transfer coefficients upstream of the throat. The plots in Figure 4 show that Aerospike 6 has lower local heat transfer coefficients upstream of the throat, up to 0.25 inches upstream of the

throat, at which point the differences in heat transfer coefficients are negligible between the two geometries. This indicates that, with everything else being equal, smoother streamlines in the converging region of the nozzle can allow for less cooling in the region upstream of the throat, which will save the cooling capacity of the cold N_2O to be used to cool the throat.

Figure 4: Local heat transfer coefficients, h , determined from CFD. The plot on the left shows the heat transfer into the spike while the plot on the right shows the heat transfer into the graphite liner on the outer wall of the nozzle.

Another design aspect investigated was the effect of the sudden expansion seen by the flow as it reaches the end of fuel grain. In this region, the boundary layer detaches at the end of the grain and there is a large recirculation zone in the wake of the fuel grain. A region such as this can cause a significant increase in heat transfer when compared to an

attached flow of similar Reynolds numbers. In order to test the hypothesis that this problem could be partially mitigated, another flow geometry was designed in order to compensate for this effect. In this case, the body of the spike was designed to extend straight back from the throat, maintaining a constant diameter in the converging region until it reached the center fuel grain. Figure 5 shows the streamlines from a CFD model of this flow at three time steps throughout the burn. It shows how the recirculation region decreases as the fuel regresses.

Figure 5: Streamlines and Mach contours for three instances in time throughout a standard burn. The prominence of the recirculation regions in the wake of the fuel grains can be seen in the initial configuration. These eddies are reduced throughout the burn.

The CFD analysis of the recirculation regions performed at RHRC also calculated the local heat transfer coefficients at the three time steps. These results are shown in Figure 6. It can be seen that while the peak heat transfer at the throat is not affected by the sudden expansion, the heat transfer into the spike as close as 0.4 inch upstream of the throat is greatly reduced as the size of the step is reduced.

Figure 6: Local heat transfer coefficients, h , determined from CFD for three instances during the course of a burn. The plot on the left shows the heat transfer into the spike while the plot on the right shows the heat transfer into the graphite liner on the outer wall of the nozzle.

After analyzing the results of these investigations, a final concept was developed for the aerospike nozzle, upon which the detailed design was based. The flow geometry of this design is shown in Figure 7. This geometry is identical to that of the cold flow nozzle starting at the throat and continuing downstream along the visible portion of the spike.

The throat area and diameter remain unchanged in keeping consistent with the specifications for a minimum length spike with the total mass flow rate at which the rocket is expected to operate.

However, there are several modifications that have been made to the flow geometry. The first is that the radius of curvature at the throat has been increased from 0.218 inch to 0.250 inch. This is motivated by the relationship between radius of curvature and local heat transfer coefficient described by Bartz⁷:

$$h_g = \frac{C}{D^{0.2}} \left(\frac{D_*}{r_c} \right)^{0.1} \left(\frac{\mu_0^{0.2} c_p}{\text{Pr}^{0.6}} \right) (\rho U)^{0.8} \left[\left(\frac{\rho_{ref}}{\rho} \right)^{0.8} \left(\frac{\mu_{ref}}{\mu_0} \right)^{0.2} \right] \quad (2.1)$$

where h_g is the local heat transfer coefficient and r_c is the radius of curvature at the throat.

Another modification is that the spike maintains a constant diameter throughout the length of the post-combustion chamber. This is meant to reduce the size of the eddy at the end of the center fuel grain early in the burn time, as well as completely eliminate it in the second half of the burn. This larger diameter section of the spike is recessed into the end of the center fuel grain in order to account for the high fuel regression rate expected in on the end of the grain.

Figure 7: The final concept for the geometry of the aerospike. The green region represents the net shape of the spike assembly. The yellow regions represent the end of inner and outer fuel grains for an annular combustion chamber.

In conjunction with the modifications to the spike geometry, the outer graphite liner geometry has been redesigned to create a smoother flow path. The radius of curvature at the throat was increased to accommodate for the increased radius of curvature on the spike. The converging surface begins further downstream than previous concepts in order to maximize the post-combustion volume in hopes of maintaining close to complete combustion inside of the rocket. The total flow area decreases monotonically, with the mean flow diameter continually decreasing throughout the post-combustion chamber all

the way up to the throat. This should smoothly accelerate the flow and thereby minimize the amount of turbulence and heat transfer.

The conceptual design of the spike assembly is shown in Figure 8. The core in the middle will most likely be made of stainless steel, with a copper sheath over it at the throat. The cooling N_2O will be fed into the stainless steel core at from the base and run up the center of the spike through a passage. It will then be routed out to the copper sheath and flow between the copper sheath and stainless steel core in several longitudinal passages of similar size to that of the Raleigh nozzle. At the base of the spike, the stainless steel core will be covered with an insulator that protects the stainless steel from being exposed to the hot flow and connects the spike to the center fuel grain. This insulator will most likely be made of graphite or a carbon-ceramic composite.

Figure 8: The conceptual design of the spike assembly. The stainless steel core is shown in gray. The insulator is shown in black. The outer sheath will be made from copper.

Future analysis will look into the detailed design of the joint between the insulator, core, and sheath. This will require a detailed thermal-mechanical finite element model to analyze the contact stresses that develop in the insulator. Another important aspect of that joint is the design of the braze joint that will hold the copper onto the stainless steel. This finite element analysis will be used to determine the best material to use for the insulator, as well as the geometry of the interface between the insulator and the core.

3. Aerospike Design Requirements

The cooled aerospike rocket nozzle being designed at Cal Poly is being built specifically for laboratory testing on the lab-scale HTPB/N₂O hybrid rocket already in place at Cal Poly. As such, many parameters in the design are driven by the combustion conditions of the laboratory rocket. The rocket combustion conditions of chamber pressure, oxidizer/fuel (O/F) ratio, and ratio of specific heats, γ , determine the spike contour and heat loads on the aerospike.

Hybrid Rocket Combustion Design Conditions

In order to allow for a wide range of testing to be performed on the aerospike, it was decided that nozzle would be designed to withstand the loading during a burn with a stoichiometric O/F ratio. In comparison to previous tests performed at Cal Poly, a stoichiometric burn will be much more aggressive with significantly higher heat loads, due to the more complete combustion of the fuel. While the aerospike nozzle will not be initially tested at these conditions, it will be designed such that it will be able to meet the heat and pressure loads that correspond with the stoichiometric combustion condition, which are the highest loads that it might see in testing.

The stoichiometric O/F ratio is 5.276. The design chamber pressure is 293.92 psia, which corresponds with a nozzle pressure ratio (NPR) of 20 at sea level. Using the NASA-Glenn Chemical Equilibrium Program (CEA),⁸ it was determined that the products of combustion have a ratio of specific heats of $\gamma=1.14$.

Aerospike Contour Design

The contour of an aerospike nozzle outside of the throat is determined based on the Prandtl-Meyer expansion fan of the exhaust gases flowing through the nozzle. The contour of a minimum length aerospike can be calculated for a specific nozzle pressure ratio and ratio of specific heats at a given operating atmospheric pressure. The contour used for the design of the aerospike was calculated using a MATLAB program that was developed as a part of a previous master's thesis at Cal Poly.⁹ Figure 9 shows the contour of the aerospike based on $NPR=20$, $\gamma=1.14$, and $P_{atm}=14.696$ psia.

Figure 9: The aerospike contour calculated for $NPR=20$ and $\gamma=1.14$ at $P_{atm}=14.696$ psia. The blue line represents the contour of the spike, while the red line shows the position of the throat of the nozzle.

Design Heat Loads

The design of a cooled aerospike rocket nozzle requires knowledge about the heat loads applied to it. Based on the geometry of the aerospike and the design combustion conditions, our research associates at Rolling Hills Research Corporation modeled the

flow of hot gas over the aerospike using FLUENT software. This computational fluid dynamics model was used to predict the heat transfer coefficients on the surface of the aerospike, assuming a constant spike surface temperature of 400°F. This is a conservative model, as the aerospike will likely reach a higher temperature at steady state and the heat flux into the aerospike will be reduced. Figure 10 shows the convective heat transfer coefficient along the length of the aerospike as computed in the FLUENT model. It shows that the highest heat transfer coefficient occurs at the throat of the nozzle. Figure 11 shows the heat load per unit length that will be transferred into the aerospike. This is calculated from the heat transfer coefficient and the exposed surface area at each axial position along the length of the aerospike, assuming a constant temperature difference between the exhaust gas stagnation temperature of $T_0=5267$ °F and the aerospike surface temperature of $T_s=400$ °F.

Figure 10: The heat transfer coefficient, h , on the surface of the aerospike as computed by the FLUENT model. The throat of the nozzle is located at $x=0$.

Figure 11: The heat load being transferred into the aerospike per unit length, calculated based on a constant surface temperature of 400 °F. This represents the amount of heat that must be transferred into the coolant at each aerospike cross-section.

The heat load shown in Figure 11 was integrated along the length of the aerospike to determine that the rate of total heat transferred into the aerospike assembly at design conditions would be 26.7 BTU/s. This is the total rate at which heat must be transferred into the coolant in order to keep the surface of the aerospike at 400 °F. This is a conservative goal for designing the cooling channel network, as it is acceptable and likely that the aerospike surface will reach temperatures above the assumed 400 °F. This would reduce the heat transfer rate from the hot exhaust gas into the aerospike in accordance with Eq. 3.1.

$$q = hA(T_0 - T_w) \quad (3.1)$$

The heat load is important because it can be used to calculate the amount of coolant mass flow required to maintain the desired temperature of the surface of the aerospike. This calculation is explained in detail in Chapter 4.

4. Idealized N₂O Coolant Process Path

The exact behavior of the N₂O coolant that flows through the aerospike is difficult to accurately model. Therefore, in order to simplify the prediction of the state of the nitrous oxide at different locations in the flow, several assumptions were made about the idealized behavior in various cases. First, it is assumed that the flow from the supply bottle through the supply lines to the rocket assembly is approximately a constant enthalpy process, resulting in just a pressure drop along the length of the lines. This constant enthalpy assumption is also applied to the injector at the entrance to the aerospike assembly that is used to adjust the pressure and total coolant flow rate through the spike, as well as the exit ports that provide a back pressure to the coolant channels. The second assumption states that the flow through the coolant channels occurs at approximately a constant pressure, with only an increase in enthalpy due to the heat transferred from the aerospike into the coolant. This idealized process path, shown in Figure 12, is used to model the coolant flow rate required such that all of the heat transferred from the hot exhaust gas into the aerospike is transferred into the coolant, heating the coolant to a quality of $x=0.5$. The design quality of $x=0.5$ was somewhat arbitrarily chosen because the liquid N₂O has a much higher heat transfer coefficient than the gaseous N₂O. More heat will be transferred into the coolant if the quality is below $x=0.5$. Designing to only heat the coolant to a quality of 0.5 also provides a margin of safety in case the coolant does not perform as well as predicted, or if the heat load turns out to be higher than predicted. If the coolant pressure in the cooling channels is assumed to be 400 psia and the supply bottle pressure is assumed to be 740 psia, the cooling capacity of the N₂O is approximately 25 BTU/lbm. This is the maximum amount of heat

that the coolant is capable of absorbing while satisfying the design condition of $x=0.5$. Based on the heat load that was calculated by integrating the heat load distribution in Figure 11, the required mass flow rate of coolant can be calculated using Eq. 4.1. This shows that in the conservative case used as a design point, the aerospike requires an approximate coolant mass flow rate of $\dot{m}=1.068$ lbm/s. This mass flow rate will be used to size the flow path and coolant channels. The flow rate will then be used to predict the convective heat transfer coefficients of the coolant.

$$\dot{m} = \frac{q_{total}}{\Delta h} \quad (4.1)$$

Figure 12: The idealized process path of the N_2O coolant flowing through the aerospike nozzle. The cooling capacity Δh represents the maximum amount of heat the coolant is capable of absorbing without the quality rising about the chosen design limit of $x=0.5$.

5. Preliminary Design Stress Calculations

In order to determine the required copper thickness and allowable channel depth, basic stress calculations were performed on the channel at the throat, where the heat load would be highest. A channel width of 0.0781 inch was selected because it could be easily manufactured using a 5/64" ball end mill. The selection of twenty 5/64" channels was confirmed by a qualitative analysis of the cooling capacity based on Reynolds number of the flow through the channel (This analysis is described in Appendix A).

Copper Shell Thickness

The dimension to be sized was the thickness of the copper shell. It was assumed that the most important load on the copper was the pressure of the coolant. It was assumed that in the worst case the pressure in the channel would be 600 psig, and if the rocket was not being fired at the time, the pressure on the outside of the copper shell would be atmospheric. The assumed 600 psig is approximately the midpoint between the expected coolant pressure in the channels of around 400 psig during operation and the supply bottle pressure, which can range from 670 psig to 850 psig depending on the temperature of the bottle. This means that the aerospike can survive a brief increase in backpressure due to the exhaust passage being plugged, but it cannot be permanently plugged and allowed to come to pressure equilibrium with the supply bottle without the copper beginning to yield.

The copper shell will be a single piece brazed to the surface of the stainless steel support material. Therefore the copper shell can be approximated as a fixed-fixed beam of infinite width with a uniform distributed load, as shown in Figure 13.

Figure 13: The load on the copper shell due to coolant pressure can be modeled as a beam of infinite width (into page) under uniform distributed loading.

Another way to describe this geometry is a flat rectangular plate that is fixed on the two long edges, with a width to span ratio that approaches infinity. This geometry is shown in Figure 14. The maximum stress that results from this loading condition can be calculated using Eq. 5.1, which can be found in *Roark's Formulas for Stress and Strain*.¹⁰

$$\sigma_{\max} = \frac{-0.5qb^2}{t^2} \quad (5.1)$$

Figure 14: The sketch of the top view of a wide beam. For the case of long coolant channels, it is assumed that the ratio of width to span is $a/b=\infty$.

A design safety factor of 2 was chosen and the copper properties were evaluated at the design operating temperature of 500 °F, which provided a yield strength of $S_y=6000$ psi. This provided an allowable stress of $\sigma_{allow}= 3000$ psi. Equation 5.1 was rearranged in order to solve for the copper thickness, resulting in Eq. 5.2.

$$t_{cu} = \sqrt{\frac{0.5qb^2}{\sigma_{allow}}} \quad (5.2)$$

For an applied load of $q=600$ psig and a channel width of 0.0781 inch, the required copper thickness was calculated to be 0.025 inch.

Channel Depth

In addition to the pressure loads applied on the copper, there will also be thermal stresses that develop due to temperature gradients and differences in coefficients of thermal expansion. It was determined that it would be reasonable to allow the copper to undergo plastic strain during the firing of the rocket, as long as the stainless steel was able to pull it back to its original configuration. Under this assumption, only the stainless steel must be prevented from yielding due to thermal strains. Therefore the channel depths were determined in order to keep the stress in the stainless steel below the yield stress. Because the channels are going to be cut radially inward from the surface of the stainless steel, the material between the channels becomes thinner as they are cut deeper.

By assuming that the copper shell would reach a temperature of 520 °F and the stainless steel core would reach a temperature of 170 °F, the resulting radii of the steel and copper were found assuming that they were allowed to grow freely. This calculation was performed using Eq. 5.3.

$$r_{hot} = (1 + \alpha \Delta T) r_{cold} \quad (5.3)$$

The distributed load, q , that the steel must exert on the inside of the copper to pull it back to the radius of the steel was calculated using Eq. 5.4 taken from *Roark's Formulas for Stress and Strain*,¹⁰ where the elastic modulus of copper, E , is assumed to be 18 Msi.

$$q = \frac{\Delta r E t_{cu}}{r^2} \quad (5.4)$$

The minimum thickness of the stainless steel ribs was determined by assuming that each rib needed to carry a concentrated load equal to the load that was distributed over its corresponding arc length. This is shown in Eq. 5.5, where θ is the angle between the ribs, σ is the stress in the rib, and t_{ss} is the minimum thickness of the stainless steel rib.

$$qr\theta = \sigma_{allow} t_{ss} \quad (5.5)$$

The yield strength of steel was evaluated at 200°F to be $S_y=18000$ psi. With a factor of safety of 2, the allowable stress was determined to be $\sigma_{allow}=9000$ psi. At the location of the throat of the nozzle the radius of the stainless steel is 0.4353 inch. For 20 channels the angle is 18° and the minimum stainless steel thickness is calculated to be 0.0539 inch. This translates to a channel depth of 0.0516 inch from the outer surface of the stainless steel to the bottom of the channel, through the analysis of the geometry of a groove cut into an 18° wedge.

6. Finalized Concept Design

The final concept design of the aerospike assembly that is being continued to a detailed design phase is shown in Figure 15. It consists of a stainless steel core, with a copper shell brazed onto the outer surface of the core. The copper shell begins approximately 1 inch upstream of the throat and continues downstream to the end of the spike. The spike is truncated to allow the coolant to flow out of the end of the aerospike assembly. The exact amount of truncation has yet to be determined, but the figure shows the aerospike being truncated to the point at which there is no more stainless steel to support the copper shell. The copper shell has a thickness of 0.025 in. The exact amount of truncation will be determined based on the advice of our research associates at Rolling Hills Research Corporation after they have performed CFD analysis on the current truncated design.

Figure 15: The final concept of the aerospike assembly, consisting of a stainless steel core covered with a copper shell and protected by a graphite or ceramic insulator upstream.

Figure 16: The stainless steel core has 20 grooves cut in it with a 5/64 inch ball end mill. Those grooves act a coolant channels. There are three different lengths of grooves: 5 long grooves, 5 medium grooves, and 10 short grooves.

The stainless steel core has 20 channels cut in it with a 5/64 inch diameter ball end mill, as can be seen in Figure 16. Five of the channels, referred to as the long channels, run the full length of the copper shell, all the way to the truncated end of the spike. The five medium channels run approximately three quarters of the length of the copper shell before being routed to an exhaust channel in the center of the spike. The long and medium channel paths can be seen in Figure 17. The remaining ten channels are used to concentrate cooling at the throat of the nozzle. These short channels can be seen in Figure 18.

The desired coolant flow rate will be obtained by properly sizing the injector that can be seen near the entrance of the stainless steel core in Figure 17. The holes that route the coolant from the center of the stainless core out to the grooves and then back to the

exhaust channel will be sized in order to balance the mass flow rates between the channels of different lengths.

Figure 17: The long channel can be seen along the upper edge of the aerospike assembly, just beneath the copper shell. The medium channel can be seen running along the bottom edge of the aerospike assembly. An injector can be seen upstream in the flow, at the entrance to the stainless steel core. The injector and the inlet and exit ports to the channels will be sized to obtain the correct coolant mass flow rate through the aerospike.

Figure 18: The short coolant channels are used to cool only throat region, where the heat load is highest. The radial channels routing the cooling to and from the outer grooves will be sized to balance the mass flow between the various channel lengths.

7. Detailed Coolant Flow Analysis

In order to analyze the flow of two-phase nitrous oxide in the aerospike, the characteristics of the flow were separated into two different types of flow. First, the frictional effects of the flow were modeled as Fanno flow. Then Rayleigh flow was used to model the effects of heating the flow. While the Fanno flow is meant to model adiabatic one-dimensional flow of a gas with friction, it is being used to model the frictional effects if heat transfer into the flow is neglected. The Rayleigh flow model assumes that the flow is frictionless but has constant heat flux into a one dimensional flow. By calculating the Fanno and Rayleigh flows separately and adding the effects together, one is able model the two-phase combined Fanno-Rayleigh flow that is occurring in the coolant channels.

The Coolant Supply State

The first step in modeling the coolant flow was to make some assumptions about the state of the coolant supply. Since the nitrous oxide is self-pressurizing and comes from a bottle that is at room temperature and the corresponding saturation pressure, a simple assumption is made that the coolant enters the supply lines as a saturated liquid at 68°F. The corresponding saturation pressure is approximately 734 psia.¹¹

The Supply Lines

Based upon the assumption that the nitrous oxide is supplied from a bottle, where it is saturated at 734 psia, the pressure drop through the supply lines was approximated. Based on data collected from hybrid rocket firings performed at Cal Poly, the pressure drop between the supply bottles and the oxidizer injectors was measured to be approximately

109 psig for an oxidizer flow rate of 0.816 lbm/s (Run #62). The coolant supply lines were assumed to have two times the losses of the main oxidizer supply lines due to differences in geometry. Since Run #62 is the motor firing with the highest nitrous oxide flow rate of 0.816 lbm/s, it was determined that it would be difficult to achieve a higher nitrous oxide flow rate for cooling the aerospike. Therefore the design N₂O coolant flow rate was selected to be 0.8 lbm/s, opposed to the more conservative 1.068 lbm/s determined from the idealized N₂O process path. This means that the quality of the two-phase fluid will be closer to 0.6 when it exits the aerospike. At this flow rate, the pressure drop in the supply lines was predicted to be 214 psig according to the following calculation.

$$\Delta P_{Supply} = 2 \left(\frac{109 \text{ psig}}{0.816 \text{ lbm/s}} \right) (0.8 \text{ lbm/s}) = 214 \text{ psig} \quad (7.1)$$

With an N₂O bottle pressure of 734 psia, this translates to a coolant pressure of 520 psia upstream of the motor.

The Inlet Pipe

In order to route the coolant from the supply line upstream of the rocket motor to the aerospike located in the center of nozzle, a coolant inlet pipe runs up the center of the inner fuel grain. This consists of a circular rod with an OD of 7/16", which supports the aerospike assembly and holds the center fuel grain in position. Up the center of this support rod runs a coolant supply channel with an ID of 0.25 inch. The supply channel is approximately 15 inches long. In order to calculate the pressure drop across the length of the supply channel, the flow of N₂O was modeled as Fanno flow. Accordingly, the drop in static pressure can be calculated as the sum of the head losses along the length of the

pipe plus the increase in dynamic pressure of the flow. This relationship is shown in the following equation.¹²

$$-dP_{Fanno} = Gdu + \frac{\rho u^2}{2} \left[\frac{4f dx}{D_H} \right] \quad (7.2)$$

where G is the mass flux, u is the velocity of the flow, f is the Fanning friction factor, x is distance in the direction of flow, and D_H is the hydraulic diameter of the channel. Due to the fact that the fluid is two-phase, the density is highly dependent on the pressure and the quality of the flow, and the quality is determined from the static enthalpy at a given pressure. The static enthalpy, h , can be related to the density according to the following equation.

$$h + \frac{G^2}{2\rho^2} = \text{constant} \quad (7.3)$$

This means that the solution to the two-phase Fanno flow problem must be found by iterating, evaluating the properties of the fluid at various pressures and qualities. This iterative solution was performed using Excel. For every possible static pressure at the downstream end of the inlet pipe, the corresponding head losses and changes in dynamic pressure were calculated. The sum of these two terms was then compared to the pressure drop across the pipe, as calculated by the following equation.

$$dP = P_{Entrance} - P_{Exit} \quad (7.4)$$

The entrance pressure was selected to be 520 psia, in accordance with the predicted losses in the supply lines. Figure 19 plots the intersection of the line representing pressure drop with the line that represents the head loss and dynamic pressure. The lines intersect at an exit pressure of 458 psia, which corresponds to a pressure drop of 62 psi.

Figure 19: The plot shows the calculated head loss and dynamics pressures at the exit of the inlet pipe for various exit pressures. The solution is found at the intersection of the pressure drop, dP , and the total losses, $HL+Gdu$. This occurs at 458 psia.

While the figure shows two intersection points at 458 psia and 186 psia, the exit pressure of 186 psia is invalid. This can be shown by repeating the calculation with a slightly higher mass flow rate. At a mass flow rate of 0.85 lbm/s through the inlet pipe, the two points of intersection occur at 442 psia and 218 psia. If the lower intersection point were valid, an increase in flow rate would result in less pressure drop, which cannot be true in this case. This conclusion is supported by the fact that the specific entropy of the fluid is higher at the 458 psia state than it would be at the 186 psia state.

Therefore the states of the nitrous oxide at the entrance and exit of the 15 inch inlet pipe were determined to be: a pressure of 520 psia and quality of 0.165 at the entrance and a pressure of 458 psia and quality of 0.204 at the exit.

The Coolant Channels

The flow of the nitrous oxide through the coolant channels in the aerospike are subject to the effects of both friction and heating. In order to determine the combined effects on the flow, Fanno and Rayleigh analyses were performed sequentially. First, the Fanno analysis of the flow was performed in the same way as was described for the coolant inlet pipe. After the state of the fluid had been determined using the Fanno analysis, that state was used as a starting point for the Rayleigh analysis. In this way the flow is analyzed as if there are two channels arranged in series, in which the first channel has coolant flowing through it adiabatically with the effects of friction and the second channel has the coolant flowing through it with heating but without the effects of friction.

There are three different lengths of coolant channels in the aerospike, as described in Chapter 6. There are five long channels that run the length of the aerospike, five medium channels that run approximately three quarters of the length of the cooled section, and ten short channels that are located at the throat of the nozzle. It was assumed that an equal amount of coolant flows through each of the twenty channels. This means that each channel has a coolant mass flow rate of 0.04 lbm/s. Furthermore, it was assumed that for any given transverse cross-section of the aerospike, the heat load on that cross-section was equally distributed between the coolant channels that are present under the copper shell in that cross-section. This means that at the throat each of the channels would be absorbing $1/20^{\text{th}}$ of the heat load, while further out along the spike the long and medium channels would each be absorbing $1/10^{\text{th}}$ of the heat load and the short channels would have no heat being transferred into them. These assumptions allowed a heat load to be developed for each type of channel.

The Long Coolant Channels

The five long channels have a total length of 1.85 inches, from the point where they branch off from the inlet pipe to where they release the coolant out of the truncated end of the aerospike. The hydraulic diameter used for the analysis is 0.05975 inch, which is the actual hydraulic diameter over most of the length of the channel. There are sections with a larger hydraulic diameter due to the transitions in shape of the channel from circular to a rounded slot, but when calculating pressure drop it is conservative to use this smaller diameter.

The Fanno analysis of the long channel shows that the nitrous oxide entering the channel at 458 psia with a static enthalpy of 92.04 BTU/lbm will exit the channel at a pressure of 432 psia and a static enthalpy of 91.99 BTU/lbm. This state was used as the starting point for the Rayleigh analysis.

Based on the previous assumptions that the heat load is evenly divided between all channels in a given cross-section and the design heat loads outlined in Chapter 3, the total heat transferred into each of the long channels is predicted to be 1.275 BTU/s, assuming the hot side copper temperature is 600°F. In a Rayleigh flow model, all of the heat added to a fluid contributes to an increase in stagnation enthalpy according to the following form of the First Law of Thermodynamics.

$$\frac{D}{Dt}(h_0) = \delta Q \quad (7.5)$$

Since the Fanno analysis showed the change in dynamic enthalpy to be negligible for the changes in velocity seen, it was assumed that all of the increase in stagnation enthalpy contributed to an increase in the static enthalpy. This increase in static enthalpy was used

to find the static enthalpy at the exit according to the following equation, where $h_{2,Fanno}$ is the static enthalpy at the exit according to the Fanno calculation.

$$h_{exit} = h_{2,Fanno} + \frac{\delta Q}{\dot{m}} \quad (7.6)$$

Accordingly, the static enthalpy at the exit was calculated to be 123.86 BTU/lbm. With this enthalpy, the quality, density, and velocity were determined for all possible exit pressures. A new change in dynamic pressure was determined for each exit pressure as well. For Rayleigh flow, Conservation of Momentum states that

$$-dP = Gdu \quad (7.7)$$

Excel was once again used to compare the pressure drop to the increase in dynamic pressure, Gdu , due to the heating of the fluid. This comparison showed that the fluid will exit the long coolant channels at a pressure of 350 psia and a quality of 0.56.

The Medium and Short Coolant Channels

The analysis described for the long coolant channels was repeated for the short and medium coolant channels. Like the long channels, the hydraulic diameter of the short and medium channels was 0.05975 inch. The length of the medium channel is 1.53 inches while the length of the short channel is 0.85 inch.

The Fanno calculations for the medium length channel found the exit pressure to be 436 psia with a static enthalpy of 92.00 BTU/lbm. The heat load applied to the medium channels is predicted to be 1.065 BTU/s. According to the Rayleigh calculations the medium channels will exit with a static pressure of 372 psia, a static enthalpy of 118.62 BTU/lbm, and a quality of 0.504.

The Fanno model for the short channels found the exit pressure to be 446 psia and the static enthalpy to be 92.01 BTU/lbm. With a heat load of 0.56 BTU/s, the Rayleigh calculations found the exit pressure to be 416 psia, with a static enthalpy of 106.05 BTU/lbm and a quality of 0.365.

The Outlet Channels

The medium and short coolant channels differ from the long channels in the fact that they do not release the exhausted coolant directly out of the truncated end of the aerospike. Instead, the coolant is routed through outlet channels from the coolant channels to the center of the aerospike where a larger exhaust port carries the coolant to the truncated end of the aerospike. These outlet channels, which run radially inward from the downstream end of the coolant channels, create more head losses that act as a back pressure to partially balance the difference in head losses between the coolant channels.

The outlet channels for the short coolant channels are 0.26 inch long and have a diameter of 0.0512 inch, which is consistent with a 1.3 mm drill bit. Based on Fanno flow calculations there will be a 20 psi pressure drop across the outlet channel, resulting in the coolant exiting at 396 psia.

The outlet channels for the medium coolant channels are 0.0895 inch long and 0.0512 inch in diameter. Fanno flow calculations show that there will be a 16 psi pressure drop across the outlet channel for the medium coolant channels resulting in an exit pressure of 356 psia.

The Coolant Process Path

The goal of this analysis was not to calculate the exact process path of the nitrous oxide as it flows through the aerospike nozzle. Instead, it was to prove that it was possible for the desired coolant mass flow rate of 0.8 lbm/s was capable of flowing through the coolant channels. Since this has been demonstrated, it will be possible to throttle back the coolant flow rate using a valve in the supply lines. This will allow for testing of the aerospike at lower coolant flow rates in order to find the limit of the coolant capacity of the nitrous oxide.

Even though the fluid states calculated in the analyses described above are not expected to the actual states of the fluid during a test firing, Figure 20 provides a visual representation of the process path that was calculated during the analysis. Figure 21, Figure 22, and Figure 23 show the details of the calculated process path for coolant flowing through the short, medium, and long channels, respectively.

Figure 20: The representation of the calculated process paths for the three different lengths of coolant channels.

Figure 21: The detailed calculated process path for flow through the short channels.

Figure 22: The detailed calculated process path for flow through the medium length channels.

Figure 23: The detailed calculated process path for flow through the long channels.

Table 1: A summary of the states of the coolant at the inlet and outlet of each region of the flow path.

	Inlet		Outlet	
Flow Region	Pressure (psia)	Enthalpy (BTU/lbm)	Pressure (psia)	Enthalpy (BTU/lbm)
Supply Lines	734	92.33	520	92.26
Inlet Pipe	520	92.26	458	92.04
Short Channels	458	92.04	416	106.05
Short Channel Outlets	416	106.05	396	106.05
Medium Channels	458	92.04	372	118.62
Medium Channel Outlets	372	118.62	356	118.62
Long Channels	458	92.04	350	123.86

8. Strategy for Finite Element Modeling of the Aerospike

The aerospike was analyzed using the several different finite element models for different regions of the assembly. All of the finite element analyses modeled both the heat transfer from the hot combustion gases to the aerospike to the N_2O coolant, as well as the stresses that develop in the materials. These analyses were performed using Abaqus CAE, run on personal computers in the Cal Poly Mechanical Engineering laboratories.

Insulator Analysis

The insulator region was analyzed using a 2D axisymmetric fully coupled thermal-mechanical model. This model covers the region beginning upstream of the aerospike, including a portion of the center fuel grain, up to the beginning edge of the copper shell. It is fully coupled in order to accurately model the gap conductance between the insulator and the stainless steel. The model includes the interactions between the insulator and the fuel grain, as well as the insulator and the stainless steel core. It was used to determine the best material to use to make the insulator. The region of the aerospike assembly that is included in this analysis can be seen in Figure 24.

Figure 24: The region shown was analyzed using a fully coupled 2D axisymmetric technique in order to determine the best material to be used as an insulator.

Throat Region Analysis

The throat region of the aerospike was analyzed using a 3D sector sequentially coupled thermal-mechanical model. This region does not require a fully coupled model because there are no gaps that can open and close and all material interfaces are brazed. The sector analyzed has an angle of 18° centered on a channel, with 9° on either side. The modeled sector is the length of the short coolant channels, covering the region from 0.3 inch upstream of the throat to 0.06 inch downstream of the throat. The model is able to analyze the effect of the curvature of the copper shell on the stress. Figure 25 shows the spike assembly up to the plane at $x=0.06$ inch, where the 3D analysis ends.

Figure 25: The 3D sector analysis of the throat section includes the curved surface of the aerospoke up to the plane where the spike is shown truncated above, at $x=0.06$ inch.

9. Finite Element Analysis Results

Two complex finite element models were developed to analyze the survivability of the aerospike. The first model was used to analyze the insulator, with several iterations used to determine a material and thickness that would meet the design requirements. The second model analyzed the throat region of aerospike in order to predict copper surface temperature and braze stresses, and to determine if the cooling capabilities of the nitrous oxide would be sufficient.

Insulator Analysis

The insulator region was analyzed using a fully coupled thermomechanical axisymmetric finite element model. The model simultaneously solved the heat transfer and stress analysis problems in the insulator region. The region was originally modeled with an insulator thickness of 0.31 inch. An insulator of this thickness was analyzed with several different materials in order to find a material that would survive the heat load of the rocket motor firing. Several materials were analyzed including graphite, Macor® ceramic, and several different brands of alumina. The graphite was found to not have a high enough tensile strength and would easily crack under the thermal load. The thermal conductivity of the Macor® was too low. This resulted in the surface temperature of the Macor® insulator reaching the maximum allowable temperature of 1832°F. The alumina has a higher thermal conductivity as well as a higher maximum allowable temperature of 3090°F. While this means that the alumina does not insulate as well as the Macor® would have, it also means that the surface temperature of the insulator will not reach the maximum allowable temperature as quickly. This means that the surface temperature of the insulator will be within an acceptable range for the first ten seconds of the burn,

which is enough time to study the cooling capacity of the nitrous oxide in the throat region. While the thermal conductivity of the alumina is higher than the Macor®, allowing more heat to be transferred inward toward the coolant, the heat path will be blocked by machining a gap between the insulator and the stainless steel core that prevents conduction between the two materials.

It was found that CoorsTek® AD-96 alumina was the material that performed the best when analyzing the original geometry. However it was found that the thermal gradient in the radial direction within the insulator caused the hoop stress on the inner surface, which is in tension, to exceed the ultimate tensile strength of the alumina. In a motor firing, this would cause the insulator to crack and most likely break into pieces, which could result in catastrophic failure of the entire rocket assembly. In order to prevent the insulator from cracking due to a steep thermal gradient, the thickness of the insulator was reduced until the model predicted that the insulator would survive the motor firing. The final insulator modeled was 0.16 inch thick.

The Model

The axisymmetric model of the insulator region consisted of four separate parts. The stainless steel core of the aerospike was modeled from the plane where the insulator ends and the copper shell begins, back upstream to include the attachment to the center support rod. The alumina insulator was located on the outer surface of the stainless steel core. The stainless steel support rod and the PMMA center grain were modeled to include two inches of material upstream of the start of the spike assembly, in order to capture the effects of the interaction between the fuel and the insulator. Figure 26 shows the axisymmetric assembly of the final geometry to be modeled.

Figure 26: The assembled parts modeled for the analysis of the insulator region. The parts are partitioned to improve the mesh quality. There are four separate parts: the spike core, the insulator, the support rod, and the center fuel grain.

The model was meshed on each part individually, with a much finer seed size being used on the insulator than the rest of the assembly, due to the fact that the insulator was the critical part being investigated with this model. In lieu of a mesh convergence study to find the converged value of stress, a mesh convergence was performed in order to achieve the correct transient thermal response to the high heat load. In this case, when a high heat load is applied to the surface of the insulator, the temperatures quickly rise at the surface, with the slope of the transient temperature response decreasing for nodes located further from the surface. However, with too course of a mesh, at the moment the

heat load is applied, the temperature at the node on the surface would quickly rise in temperature, while the node next to it, just beneath the surface would quickly decrease, often reaching negative temperatures before starting to rise around a quarter of a second into the simulation. This was unacceptable because the insulator could possibly fail due to the stresses from the transient load, so the temperatures must be correctly calculated instantaneously. Therefore, the seed size of the mesh on the insulator was decreased until the first five nodes located under the hot side surface all had increasing transient temperature responses with appropriate slopes according to their distance from the surface. Figure 29 shows the final mesh used to analyze the insulator region. All of the elements are fully coupled temperature and displacement quadrilateral elements. The insulator has 4127 elements, the spike core has 442 elements, the fuel grain has 901 elements, and the center rod has 90 elements. Figure 27 and Figure 28 show the thermal and mechanical loads that were applied to the model, respectively. Table 2 summarizes the magnitudes of those loads.

Figure 27: The thermal loads applied to the insulator region.

Figure 28: The mechanical loads and boundary conditions that were applied to the insulator region.

Table 2: The loads applied to the insulator model.

	Hot Side	Cold Side
Mechanical Load	$P_{HOT} = 300 \text{ psi}$	$P_{COLD} = 460 \text{ psi}$
Heat Transfer Coefficient	$h_{HOT} = 0.00025 \text{ BTU/in}^2\text{-s-}^\circ\text{F}$	$h_{COLD} = 0.015 \text{ BTU/in}^2\text{-s-}^\circ\text{F}$
Free Stream Temperature	$T_{HOT} = 5267 \text{ }^\circ\text{F}$	$T_{COLD} = 30 \text{ }^\circ\text{F}$

Figure 29: The meshed assembly of the insulator region.

The Results

The results of the final geometry analyzed are presented in the figures below. Figure 30 shows the deformed contour plot of the von Mises stress in the insulator region at the end of a ten second burn time. The deformed contour plot in Figure 31 shows the corresponding temperature distribution at that moment in time. The high temperatures in the fuel grain are present because in order to model the regression of the fuel, the properties of the fuel grain elements were turned off above the melting temperature of PMMA. Those elements above 350°F have negligible stiffness and thermal resistance

and therefore contribute nothing to the model once the material has melted. The four nodes highlighted in Figure 31 are used as locations for the temperature history plot shown in Figure 32. It can be seen in the temperature history plot that the surface temperature of the insulator does not exceed the maximum service temperature of the CoorsTek alumina.

Figure 30: Deformed contour plot of the stress distribution in the insulator region of the aerospike after a 10 sec burn.

Figure 31: Deformed contour plot of the temperature distribution in the insulator region of the aerospike after a 10 second burn. The four nodes highlighted show the locations from which temperature histories were taken.

Figure 32: The history plot of nodal temperatures at four important locations in the insulator region. Notice that the temperature on the surface of the insulator does not reach the maximum operating temperature of 3090°F for AD-96 CoorsTek Alumina.

Even if the insulator remains below the maximum allowable temperature of the alumina, it is critical that it does not break due to the thermal stresses that develop due to the high heat load. This would occur because the outer surface of the insulator is getting very hot and expanding much more than the inner surface of the insulator. The hot surface region and the cooler inner region must come to mechanical equilibrium. This puts the inner surface of the insulator in tension in the hoop direction as it is pulled to become larger than it would be if it was freely expanding due to the rising temperature. Meanwhile, the outer surface of the insulator is being constrained from growing as large as it would if it

were freely expanding. This puts the outer surface in compression in the hoop direction. Because the alumina is brittle and the ultimate strength is lower in tension than in compression, the inner surface of the insulator is the most critical stress location. The ultimate tensile strength of the alumina is 32100 psi. In order to maintain a safety factor of 1.2, the tensile stress must remain below 26750 psi. Figure 33 plots the maximum principal stress in the insulator at $t=0.9$ seconds with the contours bounded by the desired tensile stress limit and the ultimate tensile stress. It shows that while the stress on the inner surface will be less than the ultimate strength and the insulator shouldn't crack, the safety factor will be less than the desired value of 1.2. This suggests that the insulator may still be slightly too thick. This fact caused the final design to have an insulator thickness of 0.15 inch, instead of 0.16 inch. The node with the highest maximum principal stress on the inner surface 0.9 seconds into the run is highlighted in Figure 33. The history plot of the maximum principal stress at this node, seen in Figure 34, shows that the maximum principal stress on the inner surface of the insulator never exceeds the ultimate strength of the AD-96 alumina. Therefore the insulator will not crack due to the high temperature gradient in the radial direction.

Figure 33: The maximum principal stress in the insulator at $t=0.9$ sec into the burn. The colored contours represent maximum principal stresses that exceed the desired peak stress of 26750 psi, which corresponds to a safety factor of 1.2. The peak stress exceeds the desired limit of 26750 psi, but does not reach the ultimate strength of 32100 psi. The highlighted node on the inner surface was used as a location for the history plot.

Figure 34: The history plot above shows the maximum principal stress on the inner surface of the insulator at the node highlighted in Figure 33. The peak maximum principal stress occurs at approximately 0.9 sec into the burn when stress reaches 30092 psi, or 93.7% of the ultimate tensile strength.

Another critical stress location is the stress concentration shown in Figure 35. According to the analysis, the insulator will crack 2.2 seconds into the burn at the stress concentration where the thickness of the insulator changes. In order to prevent a crack from developing during the burn and fragments possibly breaking off and blocking the throat of the nozzle, the insulator will be manufactured in two pieces, with a thicker supporting base holding the separate constant thickness cylinder in place.

Figure 35: The maximum principal stress in the insulator at the upstream end after 2.2 seconds of burn. The stresses at the stress concentration exceed the ultimate tensile strength of alumina, which is 32100 psi. This result is the driving reason behind designing the insulator to be made of two separate pieces of alumina.

Another stress concentration occurs at the contact between the downstream end of the insulator and the stainless steel core, as shown in Figure 36. The contact between the slanted end of the insulator and the stainless steel core allows heat transfer into the stainless steel, which cools the inner surface of the alumina and creates higher stresses due to the steeper thermal gradient. In addition to this, the angle of the end surface creates a bending moment as the insulator expands and is pushed into the stainless steel. These two effects combine to cause the alumina to crack on the inner corner. In order to reduce

the stress on the inner corner of the insulator, it was determined that the end of the insulator should be stepped. This would prevent the bending moment from developing due to the angled contact surface. Also the surface area that is contacting the stainless steel will be reduced as the alumina gets hot and gaps open in the radial direction. This will result in less heat transfer into the stainless steel and a lesser temperature gradient. The stepped end of insulator will still accommodate the angled coolant passages in the stainless steel that first inspired the angled insulator.

Figure 36: Contour plot of the maximum principal stress in the downstream end of the insulator at $t=2.2$ seconds. The peak stress exceeds the ultimate tensile strength of the alumina, which inspired the decision to redesign the end to be stepped instead of tapered.

Throat Analysis

The throat region of the aerospike was analyzed using a sequentially coupled thermomechanical 3D sector model of one coolant channel. At the throat there are twenty coolant channels, making the sector is 18° wide. The model is the length of the shortest coolant channels in the throat region, starting 0.3 inch upstream of the throat and continuing to 0.06 inch downstream of the throat. The analysis modeled 3 seconds with nitrous oxide flowing through the channel, followed by five seconds with the addition of the heat load on the surface of the copper, for three different coolant heat transfer coefficients. The heat transfer coefficients are based on bounding data from the Rayleigh flow cooled nozzle experiments previously conducted at Cal Poly. The lower bound of the nitrous oxide convective heat transfer coefficient is $0.003734 \text{ BTU/in}^2\text{-s-}^\circ\text{R}$, while the upper bound is $0.032871 \text{ BTU/in}^2\text{-s-}^\circ\text{R}$. The mid-range value used to analyze the cooling at the throat was $0.014889 \text{ BTU/in}^2\text{-s-}^\circ\text{R}$.

The Model

The assembly of the model is shown in Figure 37. The copper shell is visible on top. Beneath the copper, the stainless steel wedge has a single groove cut down the length of it using a $5/64''$ ball end mill, which gives the bottom of the channel a curved contour. The end of the channel can be seen on the end of the wedge shown in the assembly. The bottom surface of the copper shell was tied to the top surface of the stainless steel webs on either side of the channel in order to simulate a brazed joint. The transient heat transfer problem was calculated first, solving for all of the nodal temperatures at each time step. Those temperatures were then used to solve a transient mechanical problem for the 3 seconds of coolant on and 5 seconds of burn time. The application of the thermal

loads is shown in Figure 38. Figure 39 shows the boundary conditions used in the mechanical model. Table 3 summarizes the magnitudes of the boundary conditions used in the thermal model. Table 4 summarized the magnitudes of the boundary conditions applied to the mechanical model.

Figure 37: The assembled geometry of the throat region that was analyzed, consisting of a copper shell brazed to a stainless steel core. The parts are partitioned to improve the mesh quality.

Figure 38: The loads and boundary conditions used for the thermal analysis of throat region of the aerospike.

Table 3: The loads used in the thermal model of the throat region. Note that $z=0$ at the throat.

	Cold Side	Hot Side $-0.3 \leq z < -0.2$	Hot Side $-0.2 \leq z < 0$	Hot Side $0 \leq z \leq 0.06$
Free Stream Temperature	$T_{\text{COOL}} = 30^{\circ}\text{F}$	$T_{\text{HOT}} = 5267^{\circ}\text{F}$	$T_{\text{HOT}} = 5267^{\circ}\text{F}$	$T_{\text{HOT}} = 5267^{\circ}\text{F}$
Heat Transfer Coefficient	Varies (Three Cases)	$h_{\text{HOT}} = 0.0011$ BTU/in ² -s-°F	$h_{\text{HOT}} = 0.002$ BTU/in ² -s-°F	$h_{\text{HOT}} = 0.00175$ BTU/in ² -s-°F

Figure 39: The mechanical loads and boundary conditions used for the stress analysis of the throat region of the aerospike. These loads are in addition to the mapping of the nodal temperatures that were calculated in the preceding thermal analysis.

Table 4: The loads applied to the mechanical model of the throat region.

	Hot Side	Cold Side
Mechanical Loads	$P_{HOT} = 300 \text{ psi}$	$P_{COOL} = 450 \text{ psi}$

The parts were meshed separately in order to accurately represent the correct geometry. The computer being used to solve the problem using Abaqus had a 32-bit Xeon quad-core processor. The computer reached the limit of its processing capabilities when it solved the mechanical problem with a mesh seed size of 0.015 inch. While it could solve the heat transfer problem with finer meshes, the simulation could not be solved when the mesh size was finer than 0.015 inch. At that seed size, hexahedral elements could not be used to accurately mesh the complex geometry of the stainless steel core. As a result, the

stainless steel core had to be meshed using poorer quality tetrahedral elements. This means that the results of the analysis could be inaccurate due to the poor quality of the mesh. The meshed assembly can be seen in Figure 40 below. The copper shell is made up of hexahedral and wedge elements, while the stainless steel core is made up of poorer quality tetrahedral elements.

Figure 40: The meshed throat model. The copper shell is made up of 18176 linear hexahedral and 576 linear wedge elements, while the stainless steel core is made up of 48047 linear tetrahedral elements.

Results

The contour plot of the temperature distribution throughout the throat region for the mid-range heat transfer coefficient is shown in Figure 41. Figure 42 shows the transient

temperature history at a node in the middle of the copper surface near the throat for all three heat transfer coefficients that were modeled. As can be seen from the plots, the upper bound and mid-range values for heat transfer coefficient predict acceptable steady state copper temperatures of approximately 510°F and 820°F, respectively, while the lower bound of the heat transfer coefficient for nitrous oxide predicts that the copper will reach an unacceptably high temperature of over 1600°F. This means that if the heat transfer coefficient for the nitrous oxide flowing through the aerospike is that low, the aerospike will overheat and begin to fail. This will happen within the first second of the motor being turned on if the motor is operating at full design performance of 294 psia chamber pressure and stoichiometric mixture ratio.

Figure 41: Deformed contour plot of the temperature in the throat region after a 5 second burn.

Figure 42: The history plot above shows the copper surface temperature at the throat as a function of time for three different values of the heat transfer coefficient, h . The units of h are $\text{BTU}/\text{in}^2\text{-s-}^\circ\text{R}$.

The stress in the stainless steel and copper was calculated for each of the three loads cases, but the results that are shown below are for the mid-range heat transfer coefficient of $h=0.014889 \text{ BTU}/\text{in}^2\text{-sec-}^\circ\text{R}$, unless otherwise noted. This is because the actual heat transfer coefficient will most likely be closest to the mid-range value. Figure 43 shows the von Mises stress distribution throughout the entire assembly, while Figure 44 shows the von Mises stress in the copper shell at the end of the simulated time period. It can be seen that the highest overall stresses occur in the stainless steel web.

Figure 43: Deformed contour plot of the von Mises stress distribution in a 3D sector analysis of a cooling channel at the throat after a 5 second burn.

Figure 44: Deformed contour plot of the von Mises stress in the copper shell after a 5 second burn.

The plastic strain in the copper shell is shown from two different viewing angles. Figure 45 shows the top surface of the copper shell, while Figure 46 shows the underside of the copper shell, where it is brazed. It should be noted that all of the copper yields during the burn, which was expected since the design relies on the stainless steel to hold the copper shell to the desired shape even as it is subjected to very high heat loads. However, as seen in Figure 46, the copper undergoes a larger amount of plastic strain on the underside, with almost 3% plastic strain being seen at the edge of the braze joint.

Figure 45: Deformed contour plot of the plastic strain in the copper shell after a 5 second burn.

Figure 46: Deformed contour plot of the plastic strain in the copper shell after a 5 second burn, viewed from the bottom. Notice that the most yielding occurs at the edge of the brazed surface.

One of the most important things to learn from this analysis is the stresses that will be subjected on the braze joint. Since the braze was modeled as a mathematical tie between the surfaces of the two materials, the braze stress is being predicted by investigating the stress in each of the two materials at the tied surfaces. Figure 47 shows the von Mises stress distribution on the underside of the copper shell. Two nodes of interest are highlighted: one at the edge of the braze joint in the upstream region, and one in the middle of the braze near the throat.

Figure 47: Deformed contour plot of the von Mises stress in the copper shell, viewed from the bottom side of the shell. There are two nodes of interest highlighted: one in the middle of the braze and one at the edge of the braze.

The stress histories for the two nodes of interest highlighted above are given in the following two plots. Figure 48 shows the history of the von Mises stress in the copper at the edge of the braze and in the middle of the braze. It can be seen that the highest stress occurs right when the motor is turned on. This event does not occur at $t=3$ seconds on this scale because Abaqus was unable to directly map the time steps between the thermal analysis and the mechanical analysis. Therefore the mechanical analysis does not have the correct time scale to match the actual motor firing. Figure 49 plots the S22 stress at the two nodes, which is of interest because it could cause the braze to “peel” open.

Figure 48: History plot of the von Mises stress at the two nodes of interest in the braze. The steep discontinuity in the stress occurs when the motor is turned on.

Figure 49: History plot of the radial stress, S22, at the two nodes of interest in the braze. The S22 stress represents the potential cause of “peel” in the brazed joint.

The results shown in the previous plots will be compared to the magnitude of the loads used to test braze samples, which will confirm whether or not braze will fail before the parent materials.

The predicted stresses in the stainless are shown in the following figures. Figure 50 shows the distribution of the axial component of stress, S_{11} . The contours show the compressive regions. The stainless steel webs are in compression because they are hotter than the stainless steel in the core of the aerospike. The webs want to grow in the axial direction, but the core material constrains it, putting it into compression. The strangely shaped red contours on the end of the model and in the core of the aerospike are caused by the poor mesh quality of the tetrahedral elements. The red contours correspond with relatively low magnitude stresses, so the accuracy of the stress calculations in the web should not be greatly affected by poor quality elements in a different region of the model.

Figure 50: Deformed contour plot of the compressive axial stress, S11, in the stainless steel core. The steel webs are in compression due to the fact that they are much hotter than the inner stainless steel, which pulls the webs into compression.

The radial stress distribution, S22, is shown in Figure 51. The red inner surface of the coolant channel is in tension because it is cooler than the rest of the stainless steel web and is being stretched by the hotter material. The blue part of the web is near the throat of the nozzle and therefore the hottest region. It is in compression because the cooler parts of the stainless steel web are preventing it from expanding further. The section cut through the stainless steel wedge reveals the poor quality of the internal tetrahedral mesh.

Figure 51: Deformed contour plot of the radial stress, S22, in the stainless steel core. The cooled surface of the web (on the sides of the channel) has the highest tensile stress, as it is being stretched by the hotter material that makes up the middle of the stainless steel webs. A section is cut away, revealing the poor quality of the internal tetrahedral meshing.

The braze stresses were also investigated on the stainless steel side of the braze. The contour plot of von Mises stress in Figure 52 has a highlighted node in the middle of the braze. That node is the location used to create the stress histories shown in Figure 53. The stress history plot shows the von Mises stress at that node for all three load cases. It is shown that the stress reaches 35 ksi when the motor is turned on for all three cases. However, as the materials yield more in the cases with less cooling, the braze stress drops.

Figure 52: Deformed contour plot of the von Mises stress in the stainless steel core. The steel located at the braze joint is one of the regions of highest stress, as expected. The high stress at the bottom of the channel was not expected and may be a result of the three dimensional nature of the problem. The highlighted node in the middle of the brazed surface atop the web on the left was used to plot a history of the stresses.

Figure 53: History plot of the von Mises stress at the node in the middle of the brazed surface, highlighted in Figure 52, for the three values of nitrous oxide heat transfer coefficient, h , given in BTU/in²-s-°R. It can be seen that the two cases of higher heat transfer coefficients, the stress in the braze remains high, while in the case of the low heat transfer coefficient the material reaches much higher temperatures and yields much more, resulting in lower stresses.

Figure 54 plots the compressive hoop stress, S_{33} , in the stainless steel core. It was expected that the stainless steel webs would be in tension, being pulled on by the hotter copper brazed to them. However it appears that the stainless steel webs are hot enough that they are being put into compression by the cooler stainless steel core. This appears to be a reasonable solution. However, the contour plot in Figure 54 shows questionable results in certain regions, which raises the question as to whether or not the tetrahedral mesh was high enough quality to solve this problem. The contours on the two webs do not match each other, even though they should be subjected to identical loads. It is possible that the entire end of the aerospike be in hoop compression because it is hotter

than the material upstream of it. However, it is strange that the highest compressive stress, shown in blue, would be on the surface of coolant channel, where the material should be cooler than the material surrounding it.

Figure 54: Contour plot of the compressive stress in the stainless steel core in the S33 direction, which represents hoop stress. The regions in compression in the S33 direction are hot and want to grow circumferentially, but are being restrained by material within the core that is cooler.

The contour plot in Figure 55 below shows the plastic strain in the stainless steel that is greater than 0.1%. The figure shows that the stainless steel will yield in the web where it is brazed to the copper, with as high as 0.5% plastic strain occurring near the braze surface.

Figure 56: The geometry that could be modeled to take advantage of another plane of symmetry in the analysis. This should cut the number of elements approximately in half.

The second thing that should be done is to improve the mesh quality. The stainless steel core should be modeled using hexahedral elements. The mesh will need to be more refined and a more powerful computer will be needed to solve the simulation. A computer cluster with 64-bit processors should be able to provide the computing power to solve a model with a more refined mesh of hexahedral elements.

The final thing that needs to be done to improve the model is to improve the boundary condition of the downstream end. The current model was solved with a free end. A better boundary condition would be generalized plane strain, that would force the end surface to remain planar while allowing the spike to grow in the axial direction. The generalized

plain strain function in Abaqus was not behaving properly when the current model was being built, so it was left out of the model.

10. Summary of the Design

A cooled annular aerospike nozzle was designed for use on a lab-scale hybrid rocket motor. The aerospike is cooled using liquid nitrous oxide, the oxidizer used in the rocket combustion, which is a novel design concept being developed at Cal Poly. The coolant is routed through a single channel inside of the support rod of the center fuel grain of the rocket motor to the nozzle. The coolant then flows through multiple parallel channels created between a stainless steel core of the aerospike and a copper shell that is brazed onto it. A ceramic insulator upstream of the throat of the nozzle prevents heat transfer from the gases in the combustion chamber into the coolant in the supply tube, thereby preserving the cooling capacity of the nitrous oxide.

The finite element analysis of the nozzle design predicted that the nozzle would survive a ten second firing of the rocket. While local plastic deformation would occur in areas of stress concentration, the net shape of the aerospike would not be changed enough to prevent the repeated firing of the aerospike. The analysis also identified several key potential failure modes that deserve further investigation as part of the development of future iterations of this design.

The first potential failure mode is cracking of the insulator caused by a steep radial thermal gradient during the transient period in the first 1 second of the rocket firing. Any changes in the insulator material or oxidizer/fuel ratio of the motor must take this failure mode into consideration, as this failure is highly dependent on the thermal capacitance, thermal conductivity, coefficient of thermal expansion, and hoop strength of the insulator, as well as heat load applied on the external surface of the insulator. A future investigation

of using a carbon/ceramic or carbon/carbon composite material for the insulator could increase the factor of safety for this failure mode. A more accurate model of the combustion chamber heat transfer at the specific rocket motor operating conditions would improve accuracy of the analysis and increase confidence in this feature of the design.

The second failure mode identified in the analysis was the surface of the insulator exceeding the maximum operating temperature of the insulator material. This would be caused by firing the rocket motor for an extended burn time significantly longer than 10 seconds. Exceeding the maximum operating temperature of the material could result in degradation of the mechanical properties or decomposition of the insulator. Risk of this failure mode can be mitigated with a more accurate prediction of the combustion chamber heat transfer rate as well as limiting the length of the motor firing time.

The third failure mode that requires more analysis is the temperature of the copper shell exceeding the operating temperature of the copper, resulting in softening, or in the extreme case melting, of the copper shell. This would result in large magnitude plastic deformation of the copper shell, preventing reuse of the aerospike. Extreme degradation of the material properties of the copper could result in the coolant channels of the aerospike starting to leak or bursting. This failure mode requires further investigation due to the fact that the lowest nitrous oxide heat transfer coefficient used in the analysis predicted this failure. This is an unlikely scenario and a highly conservative operating condition to analyze, but more accurate hot and cold side heat transfer coefficients would increase the confidence in this aspect of the design. These more accurate coefficients require a significant amount of more research and may require a much more complex

analysis which includes the thermal interaction of both the hot and cold side fluids with the copper in a single thermal model of the fluid structure interactions.

The final failure mode identified in the analysis that may require further analysis is the low cycle thermal fatigue of the copper and stainless steel at the braze joint. A more detailed model of the braze joint could be analyzed including the repeated thermal loading of multiple firings of the rocket with the aerospike nozzle. The analysis would investigate the repeated heating and cooling of the aerospike and the resulting stresses and plastic strains that develop near the braze between the copper and the stainless steel. An investigation of the strain ratcheting in this region could predict the number of times the aerospike could be used before a failure would occur due to thermal fatigue.

11. List of References

1. “Optimal Thrust Vectoring for an Annular Aerospike Nozzle,” NASA STTR Phase II Grant, Contract NND06AB82C, 06/06 – 06/08. Principal Investigator: T.W. Carpenter, co-Investigators: W.R. Murray. \$600,000: [Rolling Hills Research Corp., \$270,000; Cal Poly, \$330,000].
2. “A Reusable, Oxidizer-Cooled, Hybrid Aerospike Rocket Motor for Flight Test,” NASA STTR Phase I Grant, Contract NNX08CD33P, 01/08 - 01/09. Principal Investigator: W.R. Murray, co-Investigators: T.W. Carpenter, J.D. Mello, P. Lemieux, and M. Kerho. \$100,000.
3. “A Reusable, Oxidizer-Cooled, Hybrid Aerospike Rocket Motor for Flight Test,” NASA STTR Phase II Grant, Contract NNX10RA29C, 10/23/09 - 04/22/11. Principal Investigator: W.R. Murray, co-Investigators: J.D. Mello, P. Lemieux, and M. Kerho. \$600,000.
4. Norr, Jeff. “An Introduction to Hybrid Rockets.” 10 October 1996. University of Illinois at Urbana-Champaign. <http://dilbert.cen.uiuc.edu/soc/isds/hybrid/intro>
5. Rocket and Space Technology, <http://www.braeunig.us/space/>
6. Nelson, Lauren M., “Rayleigh Flow of Two-Phase Nitrous Oxide as a Hybrid Rocket Nozzle Coolant,” MS Thesis, California Polytechnic State University, San Luis Obispo, CA, Sept. 2009.
7. Bartz, D.R., *A Simple Equation for Rapid Estimation of Rocket Nozzle Convective Heat Transfer Coefficients*. Jet Propulsion, 1957. 27(1): p. 49-51.
8. *CEA - Chemical Equilibrium with Applications, Version 2.0*, NASA Glen Research Center, 1994.

9. Imbaratto, David, "The Interaction between Throttling and Thrust Vectoring of an Annular Aerospike Nozzle," MS Thesis, California Polytechnic State University, Sept. 2009.
10. Young, Warren C., *Roark's Formulas for Stress and Strain*. New York, NY: McGraw-Hill, 1989.
11. (NIST) *National Institute of Standards, Standard Reference Database*: <http://webbook.nist.gov/chemistry/fluid/>, U.S. Secretary of Commerce on behalf of the United States of America, 2010.
12. Zucker, Robert D. and Oscar Biblarz, *Fundamentals of Gas Dynamics*. Hoboken: Wiley, 2002.
13. Incropera, Frank P., David P. DeWitt, Theodore L. Bergman, and Adrienne S. Lavine, *Introduction to Heat Transfer*. Hoboken: Wiley, 2007.
14. White, Frank M., *Viscous Fluid Flow*. Boston: McGraw-Hill, 2006.
15. Brennen, Peter A., "Simulation of an Oxidizer-Cooled Hybrid Rocket Throat: Methodology Validation for Design of a Cooled Aerospike Nozzle," MS Thesis, California Polytechnic State University, San Luis Obispo, CA, June 2009.
16. Thornton, Earl A., and Pramote Dechaumphai. "A Taylor-Galerkin Finite Element Algorithm for Transient Nonlinear Thermal-Structural Analysis." *AIAA Journal* (1986): 210-220.
17. Nicholson, D. W., *Finite Element Analysis: Thermomechanics of Solids*. Boca Raton: CRC, 2008.

18. Cook, Robert D., David S. Malkus, Michael E. Plesha, and Robert J. Witt.
Concepts and Applications of Finite Element Analysis. New York, NY: Wiley,
2002.
19. Yeh, C. L., C. Y. Wen, Y. F. Chen, S. H. Yeh, and C. H. Wu, "An Experimental
Investigation of Thermal Contact Conductance Across Bolted Joints,"
Experimental Thermal and Fluid Sciences, Volume 25, pp. 349-357, Elsevier,
June 2001.
20. Singhal, V., Litke, P.J., Black, A.F., and Garimella, S.V., "An Experimentally
Validated Thermo-mechanical Model for the Prediction of Thermal Contact
Conductance," International Journal of Heat and Mass Transfer, Volume 48, pp.
5446-5459, Elsevier, Dec. 2005.

Appendices

A. Aerospike Cooling Channel Preliminary Analysis

In order to begin the detailed design of the cooling channels of the aerospike, an analytical analysis of the cooling capacity of various cooling channel configurations was performed. The initial assumption was that the coolant channels would consist of grooves cut into a stainless steel core with a copper shell brazed onto the outside surface. An attempt was made to find the coolant channel size that would maximize both the amount of cooling and the cooling efficiency.

Assumptions About the Cooling Model for N₂O

The cooling channel analysis was based on the assumption that the cooling capacity of nitrous oxide would be driven by the Reynolds number of the flow in the channel. This assumption can be improved upon once a complete model for the cooling with nitrous oxide is obtained. At the time being it has been assumed that a coefficient of convection for nitrous oxide is proportional to the Reynolds number based on the hydraulic diameter of the channel in accordance with Eq. A.1:

$$h \propto \text{Re}_D \quad (\text{A.1})$$

This is based on the technique used to model convective heat transfer based on Nusselt number. The heat transfer coefficient is proportional to the Nusselt number, as it is defined in Eq. A.2. The Prandtl number is a function of fluid properties, which is assumed to be relatively consistent between various geometries, since the coolant will always be a two-phase mixture of nitrous oxide. The exponent, α , which operates on the Reynolds number, is modeled differently according to different models. In the Colburn

correlation, α has a value of 0.8.¹³ In the Petrakhov correlation, α is modeled with a value of 1.¹⁴ The Petrakhov correlation was selected because it was a simpler model to work with in this situation.

$$h \propto Nu_D = C Re_D^\alpha Pr^\beta \quad (A.2)$$

The Reynolds number based on hydraulic diameter is defined according to Eq. A.3:

$$Re_D = \frac{\rho V D_H}{\mu} \quad (A.3)$$

In order to model the velocity in the cooling channel, it was assumed that for all of the designs to be compared the pressure applied upstream of the spike would be equal. Assuming that the fluid resistance through the channel could be modeled by Eq. A.4, the total volumetric flow rate of the N₂O through the spike would be modeled by Eq. A.5, where N is the number of parallel flow channels.

$$R_{EQ} \propto \frac{L}{D_H N} \quad (A.4)$$

$$Q_{Total} \propto \frac{P_{Supply}}{R_{EQ}} \propto \frac{P_{Supply} D_H N}{L} \quad (A.5)$$

From this total volumetric flow rate of the nitrous oxide, it can be found that the fluid velocity in the coolant channel behaves in accordance with Eq. A.6. Therefore the Reynolds number of the flow in a given channel is related to the geometry of the channel by Eq. A.7. Based on the assumptions that supply pressure, channel length, and fluid properties are approximately equivalent between the various designs, a model for the convection coefficient that is based solely on the channel geometry is obtained in Eq. A.8.

$$V \propto \frac{D_H}{A} \quad (\text{A.6})$$

$$\text{Re}_D \propto \frac{\rho}{\mu} \frac{D_H^2}{A} \quad (\text{A.7})$$

$$h \propto \frac{D_H^2}{A} \quad (\text{A.8})$$

As with previous cooled nozzles designed at CPSU, an initial design assumption is that all of the heat transferred into the cooling fluid is transferred from the copper and not from the stainless steel. Therefore the total heat transferred into the nitrous oxide is a function of the wetted copper surface area and the convection coefficient. For a given cross-section of the spike, the amount of heat transferred into an individual coolant channel is governed by Eq. A.9, where s is the path length along the wetted copper surface in that cross-sectional view. Equation A.10 models the total heat transferred into the coolant.

$$\dot{q}_{Channel} \propto \frac{D_H^2}{A} s \quad (\text{A.9})$$

$$\dot{q}_{Total} \propto \frac{D_H^2}{A} sN \quad (\text{A.10})$$

The most efficient cooling design will result in the most heat being transferred into a given amount of nitrous oxide flowing through the spike. By combining Eq. A.10 and Eq. A.5 with the assumption that supply pressure and channel length will be equal for different channel designs the model for cooling efficiency is obtained, as given in Eq. A.11.

$$\frac{\dot{q}_{Total}}{Q_{Total}} \propto \frac{D_H}{A} s \quad (A.11)$$

Analytical Comparison of Coolant Channel Designs

Several designs of coolant channel shapes were compared using the simplified cooling model described in the previous section. The geometries were developed based on predicted stresses that would occur during the operation of the aerospike. For each of those allowable geometries the predicted cooling capacity and cooling efficiency of the nitrous oxide was calculated, in accordance with the model. These values were then plotted in hopes of finding an optimum shape. This study was heavily based on stress analysis of the cooling channels.

The driving dimension of the coolant channel is the width. Based on a selected width of the coolant channel, the required thickness of the copper shell was calculated using Eq. A.12, which can be found in *Roark's Formulas for Stress and Strain*.¹⁰

$$\sigma_{max} = \frac{-0.5pw^2}{t_{cu}^2} \quad (A.12)$$

The pressure was assumed to be 600 psi and the required copper thickness, t_{cu} , was found such that the maximum stress did not exceed 50% of the yield strength of the copper. This is a conservative calculation as the copper shell will most likely see less than 600 psi and a safety factor of 2.0 is built into the design.

Having obtained a given copper shell thickness, it was used to calculate the allowable channel depth. Because the channels are going to be cut radially inward from the surface of the stainless steel, the material between the channels becomes thinner as they are cut deeper. The minimum thickness of this stainless steel rib is governed by the thermal

stresses that develop during the rocket firing. By assuming that the copper shell would reach a temperature of 520 °F and the stainless steel core would reach a temperature of 170 °F (based on knowledge of testing of an N₂O-cooled converging-diverging nozzle), the resulting radii of the steel and copper were found assuming that they were allowed to grow freely. The distributed load, q , that the steel must exert on the inside of the copper to pull it back to the radius of the steel was calculated using Eq. A.13, where the elastic modulus of copper, E , is assumed to be 18 Msi.

$$q = \frac{\Delta R E t_{cu}}{R^2} \quad (\text{A.13})$$

The minimum thickness of the stainless steel ribs was determined by assuming that each rib needed to carry a concentrated load equal to the load that was distributed over its corresponding arc length. This is shown in Eq. A.14, where θ is the angle between the ribs, σ is the stress in the rib, and t_{ss} is the minimum thickness of the stainless steel rib.

$$qR\theta = \sigma t_{ss} \quad (\text{A.14})$$

The minimum stainless steel thickness was found such that the stress in the stainless steel did not exceed 50% of the yield strength of the steel. This minimum stainless steel thickness was then used to find the allowable depth of a channel for each selected channel width.

Channel designs based on channels that are cut by both square and ball end mills were analyzed for channel widths ranging from 0.01 inch to 0.09 inch. The parameter expressed in Eq. A.10 that is proportional to the total heat transferred into the nitrous oxide is plotted in Figure 57. The parameter expressed in Eq. A.11 that is proportional to cooling efficiency is plotted in Figure 58.

Figure 57: The parameter plotted above is proportional to the total heat transferred into the nitrous oxide. As can be seen from the plots, a higher number of channels results in more cooling. Also, channels cut with a ball end mill can cool more than square channels of the same size.

Figure 58: The plot above shows the dimensionless parameter that is proportional to the cooling efficiency of the coolant channel designs. As can be seen above, square bottomed channels cool more efficiently than channels cut with ball end mills for a given channel width. Also, having more channels results in more efficient cooling.

An investigation of Figure 57 and Figure 58 shows that the amount of heat transferred into the N_2O and the cooling efficiency of the N_2O are not both maximized by the same

coolant channel design. Therefore an attempt was made to find a compromise between the two parameters. A new parameter was obtained by multiplying the two parameters together, as shown in Eq. A.15. This third parameter is plotted in Figure 59.

$$\dot{q}_{Total} * \frac{\dot{q}_{Total}}{Q_{Total}} = \frac{D_H^3 s^2 N}{A^2} \quad (A.15)$$

Figure 59: The parameter plotted above is used to maximize the total cooling as well as cooling efficiency. The red arrow points to a channel width of 5/64 inch, which corresponds with a ball end mill that is readily available for purchase. It is the size that maximizes the parameter without requiring custom tooling.

As shown in Figure 59, the design that maximizes both total heat transfer and cooling efficiency without requiring any custom tooling is 20 cooling channel that are cut with a 5/64" ball end mill. A representation of what the cross-section would look like at the throat is shown in Figure 60.

Figure 60: A cross-sectional slice of the spike showing 20 coolant channels cut with a 5/64" ball end mill.

B. Development of Fully Coupled Thermal/Mechanical FEA

In order to facilitate the design and testing of rocket nozzles, detailed finite element models of the nozzles are used to calculate the heat conduction in the rocket nozzle, as well as the stresses that are produced by both the thermal and mechanical loads. In previous research work at Cal Poly, the finite element analyses have used sequentially coupled thermomechanical models to analyze the rocket nozzles.¹⁵ The purpose of this section of the report is to investigate the differences between the sequentially coupled method and a fully coupled thermomechanical analysis. It investigates the advantages and disadvantages of each analysis and attempts to determine which method should be applied in various situations.

Theory of Thermomechanical Coupling

The solution of coupled thermomechanical finite element equations can be used to find the temperature and displacement fields in solids which are undergoing heat transfer. A sequentially coupled finite element solution relies on the assumption that the thermal solution to the heat transfer problem is not influenced by the mechanical solution for the displacements. The sequentially coupled model is solved by finding the temperature field first through the solution of the heat transfer problem. The temperature field found in the heat transfer problem is then copied into the mechanical problem and the displacements and stresses are determined based on these temperatures through a series of quasi-static mechanical analyses.¹⁶

A fully coupled finite element model is required in the case in which the displacements influence the solution to the heat transfer problem. In this case, the thermal equations and

displacement equations must be solved simultaneously using an iterative solution technique due to the nonlinearity of the coupling. Situations in which a physical problem must be solved using a fully coupled finite element model include heat-producing friction problems, problems with nonlinear boundary conditions, and problems that include contact in which gaps can open up.

The development of the mechanical finite element model can be based on the Principle of Virtual Work as follows¹⁷:

$$\int tr(\delta[E][S])dV + \int \delta[u]^T \rho[\ddot{u}]dV = \int \delta[u]^T [t(s)]dS \quad (B.1)$$

Where

$[E]$ = Elastic Property Matrix

ρ = Density

$[u]$ = Displacements

$[t(s)]$ = Surface Traction

$[S] = 2\mu[E] + \lambda[tr[E] - \alpha(T - T_0)][I]$

T = Temperature

α = Coefficient of Thermal Expansion

μ = Shear Modulus

λ = Lamé's First Parameter

This leads to the finite element form of the equation is as follows:

$$[K][\gamma(t)] + [M]\ddot{[\gamma(t)]} - [\Omega]^T [\theta(t)] = [f] \quad (B.2)$$

Where

$\gamma(t)$ = Nodal Displacements

$\theta(t)$ = Nodal Temperatures

$[K] = \int [B]^T [D] [B] dV$ Stiffness Matrix

$[M] = \int [N] \rho [N]^T dV$ Mass Matrix

$[\Omega] = \int \alpha \lambda [b]^T [N_r] dV$ Thermoelastic Matrix

$[f] = \int [N] [t(s)] dS$ Consistent Force Matrix

$[D]$ = Isothermal Tangent Modulus Tensor

$[b], [B]$ = Strain – Displacement Matrices

The heat transfer problem can be described by Fourier's Law for isotropic materials as follows:

$$-\nabla^T k_T \nabla T + \rho c_e \dot{T} = 0 \quad (\text{B.3})$$

This leads to the thermal field equation, which is the differential equation which will be solved to obtain the temperature field. The thermal field equation is as follows:

$$-k_T \nabla^2 T = \alpha \lambda T_0 \text{tr}[\dot{E}] + \rho c_e \dot{T} \quad (\text{B.4})$$

Where

k_T = Conductivity

c_e = Coefficient of Specific Heat

From the thermal field equation, variations can be used to find a thermal analog to the Principle of Virtual Work. This variational method yields:

$$\int \delta \nabla^T T k_T \nabla T dV + \int \delta T \rho c_e \dot{T} dV = \int \delta T \left(-[n]^T [q] \right) dS \quad (\text{B.5})$$

where $(-[n]^T [q])$ represents the heat load applied to the surface, or the thermal equivalent of surface tractions.¹⁷ Similar to the mechanical finite element theory, the heat transfer equation developed using the variational method can be reduced to something that looks like $[K]\{D\} = \{f\}$. That form is as follows:

$$[K_T][\theta(t)] + [M_T][\dot{\theta}(t)] = [f_T] \quad (\text{B.6})$$

This holds true for general conductive heat transfer equations. However, in order to express the full coupling between the strain and the temperature, the Thermoelastic Matrix, $[\Omega]$, must be included in the heat transfer equation. The resulting equation is the fully coupled thermal finite element equation:

$$\frac{1}{T_0}[K_T][\theta(t)] + \frac{1}{T_0}[M_T][\dot{\theta}(t)] + [\Omega][\dot{\gamma}(t)] = \frac{1}{T_0}[f_T] \quad (\text{B.7})$$

Where

$$\begin{aligned} [K_T] &= [K_{T1}] + [K_{T2}] \text{ Thermal Stiffness Matrix} \\ [K_{T1}] &= \int [B_T] k_T [B_T]^T dV \text{ Conductance Matrix} \\ [K_{T2}] &= \int [N_T] [n]^T [h] [N_T]^T dS \text{ Surface Conductance Matrix} \\ [M_T] &= \int [N_T] \rho c_e [N_T]^T dV \text{ Thermal Mass Matrix} \\ [f_T] &= - \int [N_T] [n]^T q_1 dS \text{ Consistent Thermal Force} \\ [h] &= \text{Surface Heat Transfer Coefficient Matrix} \end{aligned}$$

In order to solve a fully coupled thermomechanical problem, Eq. B.2 and Eq. B.7 must be solved simultaneously. This set of coupled thermomechanical equations can be written as state equations as follows¹⁷:

$$\begin{aligned}
[W_1] \frac{d}{dt} \begin{bmatrix} [\dot{\gamma}(t)] \\ [\gamma(t)] \\ [\theta(t)] \end{bmatrix} + [W_2] \begin{bmatrix} [\dot{\gamma}(t)] \\ [\gamma(t)] \\ [\theta(t)] \end{bmatrix} &= \begin{bmatrix} [f(t)] \\ 0 \\ \frac{1}{T_0} [f_T(t)] \end{bmatrix} \\
[W_1] &= \begin{bmatrix} [M] & 0 & 0 \\ 0 & [K] & 0 \\ 0 & 0 & \frac{1}{T_0} [M_T] \end{bmatrix} \\
[W_2] &= \begin{bmatrix} 0 & [K] & -[\Omega]^T \\ -[K] & 0 & 0 \\ [\Omega] & 0 & \frac{1}{T_0} [K_T] \end{bmatrix}
\end{aligned}$$

While the fully coupled thermomechanical model must solve this complete set of state equations, a sequentially coupled thermomechanical model can be solved by first solving Equation B.6 for the nodal temperatures, $[\theta(t)]$. This solution can then be plugged into Equation B.2, which is then used to solve for the nodal displacements, $[\gamma(t)]$.

All of these equations were formulated assuming that the problem is transient and the solution is a function of time. If this is true, a transient solution method such as the direct integration method would have to be used to solve the equations as a function of time.¹⁸

In a transient sequentially coupled thermomechanical analysis, the solver would have to step through time, integrating the temperatures at each time step in order to solve the heat transfer problem at discrete moments in time. The nodal temperatures at those moments in time would then be used to solve the mechanical finite element equation to get the nodal displacements at each of those moments in time.

In a transient fully coupled thermomechanical model, both the thermal and mechanical equations must be solved before the solver steps forward in time. At the first time step,

the thermal equation is solved for the nodal temperatures. Those nodal temperatures are then plugged into the mechanical equation. The mechanical equation is then solved for the nodal displacements. Those nodal displacements are subsequently used to find any modifications in the thermal stiffness matrix. Once a new thermal stiffness matrix is found, it is plugged into the thermal equation along with the nodal displacement rates, $[\dot{\gamma}(t)]$, and the nodal temperatures at the first time step are recalculated. This is repeated until the nodal temperatures converge to within a user-specified tolerance.¹⁸ This process of iteratively solving the coupled equations is repeated at each time step throughout the length of the analysis.

Analysis of an Insulator in a Rocket Nozzle

The problem chosen for investigation is the analysis of an insulated tube located in the post-combustion chamber of a rocket that carries coolant to the nozzle. For a plug style nozzle to be cooled, the coolant must be fed through a tube up the center of the rocket engine. In this case the tube is assumed to be a stainless steel rod with a 1 inch OD and a 0.5 inch ID. The stainless steel rod is insulated from the combustion gases by a shell of carbon graphite which has a 2 inch OD and a 1 inch ID. The inside of the stainless steel tube has helium flowing through it. The outside of the graphite cylinder is exposed to the combusting gases that are being accelerated toward the nozzle. This geometry is shown in Figure 61.

The temperature and heat transfer coefficients of the gases are assumed to be similar to those used in previous analyses of rocket nozzles. The helium is assumed to be at 75°F and have a convection coefficient of 0.01 BTU/in²-°R. The hot exhaust gases on the outside of the graphite shell are assumed to be at 4000°F and have a convection

coefficient of $0.001 \text{ BTU/in}^2\text{-}^\circ\text{R}$. The material properties of the graphite and the steel are based on those used by a previous graduate student in his analysis of another rocket nozzle. The nominal coefficient of conduction across the interface between the graphite and the steel was assumed to be $0.005 \text{ BTU/in}^2\text{-}^\circ\text{R}$.¹⁵ The material is assumed to be at an initial temperature of 70°F and is subjected to the hot gases for 10 seconds.

Figure 61: The geometry of the graphite shell covering the stainless steel cylinder.

Modeling the Insulator

The problem was modeled in ABAQUS as both a sequentially coupled and a fully coupled thermomechanical analysis. The part was modeled as an axisymmetric assembly. The sequentially coupled model used DCAX8 elements for the heat transfer analysis and CAX8R for the mechanical model. The DCAX8 element is an 8-node quadratic axisymmetric heat transfer quadrilateral. The CAX8R element is an 8-node biquadratic axisymmetric quadrilateral with reduced integration. The fully coupled mechanical model used the CAX8RT, which is an 8-node thermally coupled quadrilateral with biquadratic displacement, bilinear temperature, and reduced integration. An example of the mesh

used is shown in Figure 62. The cylinders were constrained at both ends in the axial direction to prevent any warping effects at a free end from influencing the results.

Figure 62: The axisymmetric model of the steel cylinder and graphite shell with a mesh size of 0.0625 in.

The analysis was run for two cases, which differed from each other based on the definitions used for the contact conductance between the two parts. The first case modeled the contact conductance as a function of the gap between the surfaces. Based on literature on the subject, it was determined that the contact conductance should be at its nominal value of $0.005 \text{ BTU/in}^2\text{-}^\circ\text{R}$ when the gap between the surfaces was zero, and should decrease to zero when the gap was greater than 0.1 inch.¹⁹

The second case that was investigated added a contact pressure dependency to the contact conductance algorithm. This case maintained the same dependency on the gap clearance as used in the first case, but added to it that the contact conductance would increase from the nominal value of $0.005 \text{ BTU/in}^2\text{-}^\circ\text{R}$ when the contact pressure was zero, up to

0.00875 BTU/in²-°R when the contact pressure reached 700 psi. This data was based on literature found that discussed the effect of contact pressure on contact conductance.²⁰

The first case was designed to yield results that should be very similar between the sequentially coupled and the fully coupled model. Because the sequentially coupled model relies on only the initial gap to determine the contact conductance, it will only yield results that are comparable to the fully coupled analysis if any gap that opens up is not large enough to greatly decrease the contact conductance across the gap. In this case the gap that opens up is very small and has a negligible effect on the contact conductance.

The second case was designed to show a situation in which the sequentially coupled analysis was inadequate. Because the heat transfer analysis is performed independent of the mechanical analysis, it is impossible for the heat transfer analysis to know what the contact pressure is at any point in time. Therefore the sequentially coupled heat transfer analysis was expected to severely deviate from the fully coupled analysis.

Results

The first case was a good example of a situation in which both the sequentially coupled and the fully coupled analyses work. The graphite was found to initially expand quickly, creating a gap between the two materials. However the gap is not very large and it does not affect the coefficient of contact conductance. This gap can be seen below in Figure 63.

Figure 63: The nodal temperature contours and displacement of the steel and graphite cylinders after 1.5 seconds. The gap begins to shrink after this time step as the steel begins to heat up.

As the heat is conducted inward through the graphite, it starts to be conducted across the interface and the steel begins to heat up. The larger coefficient of thermal expansion of the steel drives it to expand and close the gap. The final deformation and temperature distribution is shown in Figure 64. Figure 65 shows the distribution of the hoop stress after 10 seconds of heating.

Figure 64: The nodal temperature distribution in the assembly after 10 seconds.

Figure 65: The distribution of the hoop stress in the assembly after 10 seconds. Notice that the peak hoop stress occurs where the coolest part of the steel is pulled to a larger diameter by the hotter material outside of it.

A mesh convergence study was performed on the results of the first case in order to determine the differences in the convergence of the two different analysis types. The convergence study was performed on both the hoop stress and the nodal temperature evaluated at the node at the lower right corner of the mesh, on the outer surface of the graphite, at the end of the 10 seconds of analysis. Figure 66 shows the temperature convergence while Figure 67 shows the convergence of the hoop stress.

Figure 66: Convergence study of the temperature on the outer surface of the graphite after 10 seconds.

Figure 67: Convergence study of the hoop stress on the outer surface of the graphite after 10 seconds.

As can be seen from the convergence studies, the temperature appears to converge much more quickly for the sequential coupling than for the full coupling. This is most likely due to the fact that the sequential analysis used quadratic heat transfer elements while the fully coupled elements only had bilinear mapping of the temperatures. On the other hand, the hoop stress appears to converge at an equal rate between the full and sequentially coupled models. This implies that quick convergence of the temperatures does not necessarily lead to quick convergence of the stress.

The best way to see the difference between the sequential and fully coupled analyses is to look at the interface between the two materials. Figure 68 shows the size of the gap between the steel and the graphite as a function of time, which is the same for both case 1 and case 2, since pressure dependency of contact conductance does not come into play at a time when there is a gap. In the figure one can see that the sequentially coupled analysis predicts the same gap growth and decrease as the fully coupled analysis. The only differences in the results are due to the fact that the sequential analysis took larger

increments in its analysis because it was a less complex problem and therefore has a more piecewise result.

Figure 68: Plot of the gap size as a function of time for various mesh sizes for the fully coupled analysis, as well as one mesh size for the sequential analysis.

The temperature across the material interface is the clearest representation of the differences between the two cases. Figure 69 shows the temperature across the interface for the first case, while Figure 70 shows the temperature difference for the second case. As can be seen from the figures, the sequential analysis can accurately describe the temperature difference when the contact conductance is very close to constant, but if the contact conductance is dependent on pressure like it is in case 2, the sequential coupling method becomes very inaccurate.

Figure 69: The temperature difference at the contact between the steel and the graphite for case 1.

Figure 70: The temperature difference at the contact between the steel and the graphite for case 2.

Conclusions

Based on the results of this study, it can be concluded that while sequential coupling works well in some cases, there are situations, such as the analysis of this region of the aerospike nozzle, in which fully coupled thermomechanical models must be used.

Sequential coupling works well in cases in which there is only one part or in cases in which the thermal interface between two parts is not strongly influenced by any gaps that open up between them. In these situations sequential coupling has the possibility to be a more efficient computational tool because higher order thermal elements can be used and the solver has fewer equations to solve simultaneously. If an interface between parts is greatly influenced by the size of the gap, or if the contact conductance is a function of the contact pressure, it is necessary to solve the fully coupled thermomechanical finite element equations.

From this analysis it has been determined that fully coupled thermomechanical finite element models should be used whenever possible for the modeling of the aerospike. This is because the design includes an insulator protecting the spike upstream of the throat. As long as the insulator is not perfectly bonded onto the stainless steel core, gaps can open up between the insulator and the stainless steel that a sequentially coupled simulation would not be able accurately model.