DEVELOPMENT OF MIRROR FLEXURES FOR USE IN THE MUVI INSTRUMENT

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

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

by
Colin W. Harrop
April 2023
COMMITTEE MEMBERSHIP

TITLE: Development of Mirror Flexures for use in the MUVI Instrument

AUTHOR: Colin W. Harrop

DATE SUBMITTED: April 2023

COMMITTEE CHAIR: Eltahry Elghandour, Ph.D.
Professor of Mechanical Engineering

COMMITTEE MEMBER: Ashraf Elbarbary, Ph.D.
Assistant Professor of Mechanical Engineering

COMMITTEE MEMBER: Mohammad Noori, Ph.D.
Professor of Mechanical Engineering

COMMITTEE MEMBER: Kodi Rider
UC Berkeley Space Sciences Laboratory
Senior Aerospace Engineer
ABSTRACT

Development of Mirror Flexures for use in the MUVI Instrument

Colin W. Harrop

The Miniaturized Ultraviolet Imager (MUVI), is a compact wide field UV imaging instrument in development at UC Berkeley Space Sciences Laboratory and Cal Poly, San Luis Obispo. MUVI is designed to fit in a 2U CubeSat form factor and provide wide field, high resolution images of the ionosphere at far ultraviolet wavelengths. This thesis details the design and analyses of MUVI’s deployable cover mirror mounting flexures. Three different flexure geometries were evaluated, an optimal candidate was determined based on a number of criteria including isolation of vibration and stress to the mirrors, manufacturability and cost. The design of the flexure system includes the flexure blades themselves, Invar pads bonded to the mirror to mitigate the difference in CTEs of the different material, mounting of flexure blades to the deployable cover and ground support equipment for assembly and testing.

During the design of the flexures, various materials were studied, and Titanium was concluded as the optimum material due to its combination of high strength and flexibility compared to stainless steel, aluminum and other metals. Utilizing titanium, several flexure designs were proposed, and three candidates were selected to be manufactured and tested. Throughout the design phase, all flexures went through several rounds of analysis utilizing finite element analysis to simulate quasi-static loads, modal analysis of the systems natural frequency as well as random vibration simulations to simulate testing environments.

Once the front-runner designs were selected and manufactured, several tests were conducted. Testing included adhesive bond coupon testing of the adhesive in tension and bending to experimentally validate
the bonding size of the invar pads would be sufficient. The adhesive bond testing conducted tension and three-point bend tests to characterize the epoxy adhesive used in the flexure assembly. Testing also consisted of sine sweep and random vibration environment in accordance with the NASA General Environmental Verification Standard to qualify the hardware for spaceflight. Throughout the vibration testing, an autocollimator was used pre and post-test to measure shifts in the optical alignment of the mirror after it underwent vibration qualification testing.

Experimental and analytical models were compared once all testing was completed. The Curved Blade showed to test in the real world very close to that predicted by the finite element model, however, the Bent Blade and Z Blade showed a larger difference between analysis and test. Discussion into the reasoning for this difference and lessons learned is included.

Keywords: Ultraviolet, Imager, Space, Vibration
ACKNOWLEDGMENTS

I would like to express my greatest appreciation to my thesis advisors, Dr. Eltahry Elghandour and Kodi Rider of UC Berkeley Space Sciences Lab for providing me the opportunity and challenge to work on this project. It was a challenging project, and it was their support and belief in me that motivated me throughout my time as a graduate student to never give up and keep pushing forward. I will never forget their undying support throughout my time as a graduate student. It was Dr. Eltahry Elghandour who first encouraged me to pursue graduate studies at Cal Poly and work on this project. Your continued support through the entire project was more than I could have ever asked for and is greatly appreciated. Thank you to Kodi Rider, who first mentored me though my senior project continued to push me to be the best engineer I could be. His commitment to this project and my success as I developed as an engineer was more than I could have ever imagined, and I am incredibly thankful.

I would also like to thank Dr. Thomas Immel of UC Berkeley Space Sciences Lab for the support through this project. To Dr. Muhammad Noori and Dr. Elbarbary, thank you for your time and support reviewing and providing feedback on my thesis. You helped me make this the best work it could be. To my family, thank you for the continued support as I completed this project.

This work was supported by the NASA Science Mission Directorate, Heliophysics Division through contract 80NSSC19K0494.
# TABLE OF CONTENTS

LIST OF TABLES.................................................................................................................................................. ix
LIST OF FIGURES ................................................................................................................................................ x

Chapter 1 – Introduction ........................................................................................................................................... 1
  Background on the MUVI Instrument .................................................................................................................. 1
  Background Research .......................................................................................................................................... 4
  Objectives and Thesis Scope .............................................................................................................................. 6

Chapter 2 – Development of an Optimum Flexure .............................................................................................. 10
  2.1 – Top Mirror Flexure Blades ....................................................................................................................... 10
  2.2 – Bottom Mirror Flexure Blades .................................................................................................................. 16
  2.3 – Assembly Fixtures ..................................................................................................................................... 18
  2.4 – Vibration Testing Mounting Fixture ......................................................................................................... 19

Chapter 3 – Analysis .............................................................................................................................................. 21
  3.1 - Mirror Bonding Analysis .......................................................................................................................... 21
  3.2 – Precursor Analysis - Cantilever Beam FEA .............................................................................................. 25
  3.3 - Flexure Blade Analysis ............................................................................................................................ 26
    3.3.1 - Initial Finite Element Models ............................................................................................................. 29
    3.3.2 – Top Mirror Flexure Analysis ............................................................................................................. 30
    3.3.3 - Bottom Mirror Flexure Analysis ......................................................................................................... 41
    3.3.4 - Further developing the Finite Element Models .................................................................................. 50
  3.4 – Glue Test Specimen FEM ........................................................................................................................ 54

Chapter 4 – Testing .............................................................................................................................................. 58
  4.1 - Testing Overview ....................................................................................................................................... 58
  4.2 - Epoxy Coupon Testing ............................................................................................................................... 58
  4.3 - Invar Plug to Mirror Bonding ................................................................................................................... 68
  4.4 - Vibration Testing ....................................................................................................................................... 73
    4.4.1 - Vibration Testing Methods .................................................................................................................. 74
    4.4.2 - Fixture Vibration Testing ..................................................................................................................... 76
    4.4.3 - Autocollimator Testing ....................................................................................................................... 79
    4.4.4 - Curved Blade Vibration Testing ........................................................................................................... 81
    4.4.5 - Bent Blade Vibration Testing ............................................................................................................... 87
4.4.6 - Z Blade Vibration Testing ............................................................... 92
Chapter 5 - Conclusion .............................................................................. 98
  5.1 - Lessons Learned ............................................................................ 104
  5.2 - Future for MUVI ............................................................................ 109
BIBLIOGRAPHY ......................................................................................... 110
APPENDICES ............................................................................................ 112
  A. Mechanical Drawing Package.......................................................... 112
  B. Bill of Materials ............................................................................. 125
  C. Vibration Test Procedure ................................................................. 126
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Table 1.</strong> Shear Stress due to thermal effects in the bond.</td>
<td>23</td>
</tr>
<tr>
<td><strong>Table 2.</strong> Comparison of numerical and analytical methods on a cantilever beam.</td>
<td>26</td>
</tr>
<tr>
<td><strong>Table 3.</strong> Cantilever beam example of mesh convergence.</td>
<td>28</td>
</tr>
<tr>
<td><strong>Table 4.</strong> Bent Blade Handling/Assembly Static Stress (44.5N load)</td>
<td>32</td>
</tr>
<tr>
<td><strong>Table 5.</strong> Bent Blade Modal Analysis Results.</td>
<td>34</td>
</tr>
<tr>
<td><strong>Table 6.</strong> Curved Blade Handling/Assembly Static Stress (44.5N load)</td>
<td>36</td>
</tr>
<tr>
<td><strong>Table 7.</strong> Curved Blade Modal Analysis Results.</td>
<td>38</td>
</tr>
<tr>
<td><strong>Table 8.</strong> Z Blade Handling/Assembly Static Stress (44.5N load)</td>
<td>40</td>
</tr>
<tr>
<td><strong>Table 9.</strong> Z Blade Modal Analysis Results.</td>
<td>41</td>
</tr>
<tr>
<td><strong>Table 10.</strong> Bottom Mirror Bent Blade Handling/Assembly Static Stress (44.5N load)</td>
<td>43</td>
</tr>
<tr>
<td><strong>Table 11.</strong> Bottom Mirror Bent Blade Modal Analysis Results.</td>
<td>44</td>
</tr>
<tr>
<td><strong>Table 12.</strong> Bottom Mirror Curved Blade Handling/Assembly Static Stress (44.5N load)</td>
<td>46</td>
</tr>
<tr>
<td><strong>Table 13.</strong> Bottom Mirror Curved Blade Modal Analysis Results.</td>
<td>47</td>
</tr>
<tr>
<td><strong>Table 14.</strong> Bottom Mirror Z Blade Handling/Assembly Static Stress (44.5N load)</td>
<td>49</td>
</tr>
<tr>
<td><strong>Table 15.</strong> Bottom Mirror Z Blade Modal Analysis Results.</td>
<td>50</td>
</tr>
<tr>
<td><strong>Table 16.</strong> Material properties used for mirror-flexure FEM.</td>
<td>51</td>
</tr>
<tr>
<td><strong>Table 17.</strong> Material Properties for Epoxy Specimen testing.</td>
<td>55</td>
</tr>
<tr>
<td><strong>Table 18.</strong> Failure point of bend test specimens for epoxy bond coupon testing.</td>
<td>65</td>
</tr>
<tr>
<td><strong>Table 19.</strong> Failure point of tensile test specimens for epoxy bond coupon testing.</td>
<td>68</td>
</tr>
<tr>
<td><strong>Table 20.</strong> Accelerometers used for vibration testing.</td>
<td>75</td>
</tr>
<tr>
<td><strong>Table 21.</strong> Curved Blade Natural Frequency Comparison.</td>
<td>87</td>
</tr>
<tr>
<td><strong>Table 22.</strong> Bent Blade Natural Frequency Comparison.</td>
<td>91</td>
</tr>
<tr>
<td><strong>Table 23.</strong> Top Mirror Z Blade comparison of analytical and experimental results.</td>
<td>96</td>
</tr>
</tbody>
</table>
Table 24. Top Mirror Bent Blade comparison of analytical and experimental results. ......................... 101
Table 25. Top Mirror Curved Blade comparison of analytical and experimental results. .................... 101
Table 26. Top Mirror Z Blade comparison of analytical and experimental results. .............................. 101
Table 27. Autocollimator Results. ........................................................................................................ 101
Table 28. Z Blade Buckling Analysis ...................................................................................................... 103
LIST OF FIGURES

Figure

**Figure 1.** MUVI Instrument in the door closed configuration. ................................................................. 2

**Figure 2.** Exploded view of the MUVI Instrument, with each senior project highlighted. ......................... 3

**Figure 3.** Deployable Cover of the MUVI instrument with original mirror mount design ....................... 3

**Figure 4.** Top Door design envelope ....................................................................................................... 11

**Figure 5.** Top Mirror Curved Flexure Blade design .................................................................................. 12

**Figure 6.** Top Mirror Bent Flexure Blade design ...................................................................................... 12

**Figure 7.** Top Mirror "Z" Flexure Blade design ......................................................................................... 13

**Figure 8.** Invar bond pad design ............................................................................................................. 14

**Figure 9.** Top Mirror with Z Flexure Blades, isometric and top views .................................................... 15

**Figure 10.** Bottom Mirror Bent Blade Flexure Design ............................................................................... 16

**Figure 11.** Bottom Mirror Curved Blade Flexure Design ........................................................................ 16

**Figure 12.** Bottom Mirror Z Blade Flexure Design ............................................................................... 17

**Figure 13.** Bottom mirror invar bond pad ............................................................................................... 17

**Figure 14.** Bottom Mirror with Bend Flexure Blades ............................................................................... 18

**Figure 15.** Top Mirror Assembly Fixture ............................................................................................... 19

**Figure 16.** Vibration Test Fixture ........................................................................................................... 20

**Figure 17.** Horizontal Shake Table for X and Y axes (left) and Vertical Shake Table for Z axis (right) . 20

**Figure 18.** Mesh convergence of cantilever beam. Coarse mesh (10mm element size) top left, medium mesh (5mm element size) top right, fine mesh (2.5 mm element size) bottom left and very fine mesh (1.25mm element size) bottom right. ..................................................................................................................... 28

**Figure 19.** Deflection plot from mesh convergence study on a cantilever beam. Coarse mesh (top left), medium mesh (top right), fine mesh (bottom left) and very fine mesh (bottom right). Deflection values in Table 3. .................................................................................................................................................. 29
Figure 20. Bent Blade mesh in SolidWorks.......................... 30
Figure 21. Bent Blade Z-axis load case ................................................. 31
Figure 22. Bent Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center)............................................................... 32
Figure 23. Bent Blade Modal Analysis with high deformation for visualization. Horizontal axes (bottom) and vertical axis (top)................................................................. 34
Figure 24. Curved Blade Assembly mesh in SolidWorks............................. 35
Figure 25. Bent Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center)........................................................................ 36
Figure 26. Curved Blade Modal Analysis with high deformation for visualization. Horizontal axes (top) and vertical axis (bottom). ................................................................. 37
Figure 27. Z Blade Assembly mesh in SolidWorks......................................... 38
Figure 28. Z Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center). ...................................................................................... 39
Figure 29. Z Blade Modal Analysis with high deformation for visualization. Horizontal axes (top) and vertical axis (bottom). ...................................................................................... 40
Figure 30. Bottom Mirror Bent Flexure Blade Assembly mesh in SolidWorks. ........................................... 42
Figure 31. Bottom Mirror Bent Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center)........................................................................ 42
Figure 32. Bottom Mirror Bent Blade Modal Analysis with high deformation for visualization. Mode 1, normal to mirror (top left), rotation about mirror plane mode 2 (top right) and mode 3 (bottom center). 43
Figure 33. Bottom Mirror Curved Flexure Blade Assembly mesh in SolidWorks........................................... 45
Figure 34. Bottom Mirror Curved Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center)........................................................................ 45
Figure 35. Bottom Mirror Curved Blade Modal Analysis with high deformation for visualization. Mode 1, normal to mirror (bottom center), rotation about mirror plane mode 2 (top right) and mode 3 (top left). ...........................................................46

Figure 36. Bottom Mirror Z Flexure Blade Assembly mesh in SolidWorks. ........................................48

Figure 37. Bottom Mirror Z Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center). ..........................................................48

Figure 38. Bottom Mirror Z Blade Modal Analysis with high deformation for visualization. Mode 1, normal to mirror (bottom center), rotation about mirror plane mode 2 (top right) and mode 3 (top left). 49

Figure 39. Top Mirror Bent Blade FEM in Abaqus........................................................................51

Figure 40. Illustrative example of a C3D8 element type [24]............................................................52

Figure 41. Top Mirror Bent Blade mesh..........................................................................................52

Figure 42. Bent Blade Flexure random response study ................................................................53

Figure 43. Three-Point Bend Adhesive Test Specimen ..................................................................54

Figure 44. Tension Adhesive Test Specimen..................................................................................55

Figure 45. FEA results of the tension test model. ..........................................................................56

Figure 46. Schematic of bonding setup to ensure consistent bond gap..........................................59

Figure 47. Bend test coupons for epoxy bond testing ....................................................................60

Figure 48. Tensile test coupons for epoxy bond testing ..................................................................60

Figure 49. AMETEK LD50 Tensile Testing machine .......................................................................61

Figure 50. Three-Point Bend Test setup .......................................................................................61

Figure 51. Bend test epoxy bond specimens after testing ...............................................................62

Figure 52. Force vs Displacement results for the bend test of epoxy bond coupons .......................63

Figure 53. Average Force vs Displacement results of the bend tests for epoxy bond coupon testing ......64

Figure 54. Tensile test setup for epoxy bond coupon testing .........................................................66

Figure 55. Tensile test coupon after failure ...................................................................................66

Figure 56. Tensile test results of epoxy bond coupon testing .......................................................67
Figure 57. Average Displacement vs Load plot for Tensile testing of epoxy bond coupons.................67
Figure 58. Bottom mirror (left) and Top mirror (right) bonding fixtures. .................................69
Figure 59. Bottom mirror with overfilled epoxy bond...............................................................70
Figure 60. Bottom mirror and Delrin ring after first bond attempt. ............................................71
Figure 61. Bottom mirror bonding fixture on the second attempt with plastic seal....................71
Figure 62. Second bottom mirror bonded having used plastic seal. ............................................72
Figure 63. Petro Wax mounting method (top left), Super Glue and Kapton Tape method (top right), and double sided Kapton Tape (bottom center).......................................................................................76
Figure 64. Empty Mounting Fixture on shake table for preliminary testing.................................77
Figure 65. Vibration Mounting Fixture sine sweep in the horizontal axis....................................77
Figure 66. Vibration Mounting Fixture sine sweep in the vertical axis........................................78
Figure 67. Autocollimator Diagram [27].....................................................................................79
Figure 68. Autocollimator Mounting Setup..................................................................................80
Figure 69. Autocollimator eyepiece after initial calibration .......................................................81
Figure 70. Vertical Shake Table test setup....................................................................................82
Figure 71. Curved Blade Flexure Vertical Axis Sine Sweep - pre and post overlay ......................82
Figure 72. Curved Blade Flexure Vertical Axis Random Vibration Input ....................................83
Figure 73. Horizontal Shake Table test setup..............................................................................84
Figure 74. Pre and Post Sine Sweep Horizontal X Axis - Curved Blade Flexure .........................84
Figure 75. Random Vibration Horizontal X Axis - Curved Blade Flexure ..................................85
Figure 76. Pre and Post Sine Sweep Horizontal Y Axis - Curved Blade Flexure .........................86
Figure 77. Random Vibration Horizontal Y Axis - Curved Blade Flexure ..................................86
Figure 78. Pre and Post Sine Sweep Vertical Axis - Bent Blade Flexure .....................................88
Figure 79. Random Vibration Vertical Axis - Bent Blade Flexure ...............................................88
Figure 80. Pre-Test Sine Sweep Horizontal X Axis - Bent Blade Flexure ....................................90
Figure 81. Random Vibration Horizontal X Axis - Bent Blade Flexure .......................................90
Figure 82. Bent Blade testing failure in X-axis.........................................................................................91
Figure 83. Pre and Post Sine Sweep Vertical Axis - Z Blade Flexure.......................................................92
Figure 84. Random Vibration Vertical Axis - Z Blade Flexure ................................................................93
Figure 85. Pre and Post sine sweep Horizontal X Axis - Z Blade Flexure ..............................................94
Figure 86. Random Vibration Horizontal X Axis - Z Blade Flexure ........................................................94
Figure 87. Z Blade Flexure Testing Failure ............................................................................................95
Figure 88. Pre-test Sine Sweep Horizontal Y Axis - Z Blade Flexure ......................................................95
Figure 89. Random Vibration Horizontal Y Axis - Z Blade Flexure ........................................................96
Figure 90. Falcon 9 Random Vibration Environment [28] ....................................................................98
Figure 91. NASA GEVS Random Vibration Qualification curve .............................................................99
Figure 92. Falcon 9 maximum random vibration predicted environment ..............................................99
Figure 93. GEVS random vibration qualification and acceptance profiles ............................................100
Figure 94. Autocollimator results - Curved Blade: X-axis, Y-axis, Z-axis (right to left) .......................102
Figure 95. Z Blade Flexure failure in Y-Axis Random Vibration testing .................................................104
Figure 96. Interference fit of Flexure Blade and Invar Bonding Pad .......................................................106
Figure 97. Comparison of fastener clearance in the flexure through hole. Standard M2.5 fastener (left)
custom shoulder bolt (right) ..................................................................................................................107
Figure 98. Illustrative image of epoxy staking fasteners ........................................................................108
Chapter 1 – Introduction
Background on the MUVI Instrument

The ionosphere is a region of the atmosphere that is of great interest to researchers at NASA and the University of California, Berkeley Space Sciences Lab (SSL). Understanding the interactions of solar radiation and “space weather” and how it interacts with Earth’s weather systems has wide ranging applications to space based technologies such as communication satellites and GPS systems [1]. In 2019 the NASA Explorer Mission ICON was launched into the ionosphere to collect data on ultraviolet light in the upper atmosphere. One of the instruments on the (Ionospheric Connection Explorer) ICON satellite was a Far Ultraviolet (FUV) imager specifically designed to image UV emissions in the Ionosphere. The Miniaturized Ultraviolet Imager (MUVI) is a NASA technology development grant to further refine and miniaturize the FUV instrument on ICON and expand on the capability with a small, relatively simple design tuned to the specific purpose of UV imaging. The UV spectrum is of particular interest to study in the ionosphere because the ionosphere has dense plasma that emits UV light. Furthermore, the densest plasma in the ionosphere is in the equatorial ionospheric anomaly (EIA), which glows brightly in UV wavelengths, in rough proportion to the density of the dominant ionospheric ions making it a great candidate to study the interactions between space weather and Earth’s atmosphere [2].

In 2018 Dr. Thomas Immel and Kodi Rider at UCB SSL and Dr. Eltahry Elghandour at Cal Poly, SLO received a NASA grant to develop the MUVI project. The goal of MUVI is to take the capabilities of ICON’s FUV instrument and bring it down to the size of a 2U CubeSat form factor of 100mm x 100mm x 227mm. So far MUVI has completed several milestones in the development of the instrument. The camera, lens assembly, deployable cover and mirrors and instrument frame have been completed as well as the high voltage power systems, the thermal analysis and design as well as the mechanical flexures of this project are still ongoing. This work has been completed by four previous Cal Poly senior project teams as well as four Cal Poly graduate thesis projects. The first senior project team designed the lens
assembly [3] followed by the second team that designed the deployable cover and front optics [4] that this project integrates with. The third senior project focused on the overall 2U CubeSat frame [5] and the fourth senior project worked to design the high voltage power systems for the MUVI instrument [6]. Jason Grillo from the first senior project team completed his graduate thesis designing a vacuum compatible testing apparatus to test and validate MUVI in a thermal vacuum chamber [7]. The second graduate thesis was completed by Elena to complete the design of the high-voltage power supply started by the senior project team [8]. The final two graduate thesis projects that are still in work are this one for the mechanical flexures and Erik Soldenwagner who is focusing on the thermal aspects of MUVI.

![Figure 1. MUVI Instrument in the door closed configuration.](image-url)
Figure 2. Exploded view of the MUVI Instrument, with each senior project highlighted.

Figure 3. Deployable Cover of the MUVI instrument with original mirror mount design
Background Research
Utilizing mechanical flexures to isolate optical elements in space-based imaging applications is a common way to solve the problem of isolating sensitive optics from the harsh environments both of launch and of space while in use. The ICON mission that is the inspiration for MUVI utilized flexures to mount the mirror in its FUV instrument [9]. The Far Ultra-Violet Imager on the Icon Mission by Mende, et al. outlines the mechanical design for the FUV instrument on ICON. Flexures on the instrument are used to isolate the mirror from the rest of the structure. The design utilizes bipod shaped thin flexures that bond to the rectangular mirror on three sides using circular bond pads and an epoxy to adhere the flexures to the mirrors. Inspiration from the ICON FUV mechanism was drawn upon when designing the flexures for the MUVI front optic. The ICON FUV presents a potential design solution direction, but as the application of MUVI and the size of the optical elements vary by a large amount, the detailed design of MUVI had the potential to take shape in unique ways.

Field Guide to Optomechanical Design and Analysis by K. Schwerz and J Burge [10] outlines a general design road map when designing optomechanical assemblies. The guide discusses such considerations as material selection, material properties of interest and other factors during initial design. The consideration of having a high strength to elastic modulus in a flexure material are discussed, as having “The greatest compliance, given the same length flexure, is achieved by the material with the greatest reduced tensile modulus, defined as the ratio of the yield strength $\sigma_{ys}$ of the material to Young’s modulus E. The higher the reduced tensile modulus is, the more desirable the material for use as a flexure.” Titanium is a clear winner as it’s strength to elastic modulus ratio is 7.27 E-3, with the only commonly available material that comes close is Beryllium Copper at 7.14E-3. Other common materials like stainless steel and aluminum are almost half that of Titanium. As Titanium is a readily available material with extensive flight heritage in various optomechanical space-based instruments, it was strongly considered when designing the flexures for MUVI.
The ICON FUV has flight heritage to show that adhesive bonding mirrors is a suitable method and the Field Guide presents the rationale for why to use adhesives in an optomechanical design. “Adhesives are useful for mounting and bonding optical and mechanical components; compared to glass and metal, they typically have high Poisson ratios, low stiffness, and much higher CTE values. Using an adhesive is a relatively quick, simple, and inexpensive mounting solution commonly used in optomechanics.” Given the large differences in CTE between that of the Fused Silica mirror, and Titanium flexures, an adhesive appears to be an optimal solution to bridge the gap as well as a simple solution to attach the mirror to the metallic structure.

*Opto-Mechanical Systems Design* [11] by Paul Yoder Jr., an expert in the field of optomechanical systems, presents more depth into using adhesives in an optical assembly. In the textbook, in depth analytical methods of calculating the stress induced into a mirror at a bonded location due to differences in the coefficient of thermal expansion between the optical element and the metallic support structure. The section goes on to use an example of a fused silica mirror bonded to a titanium base showing that over a 90 °F temperature change, there is enough stress induced due to differences in CTE that the mirror can catastrophically fracture. The section goes on to present a rule of thumb for stress in bonds as well as a potential method to counteract this failure: “This stress significantly exceeded the 1000 lb/in.2 rule-of-thumb tolerance established earlier for tensile stress in glass, so breakage of the prism was to be expected at the specified low temperature. As shown in the photograph, failure did indeed happen. It was suggested that a bond area comprising three equal circles of 0.250 in. diameter could replace the larger single bond.”

An alternative method to counteract the large CTE differences, or rather, a method to use in conjunction with smaller bonding areas, is to use a metal with a CTE closer to that of the fused silica as a bridge between the titanium flexure and the mirror. This suggests that Invar would be a good material to investigate as it is one of the lowest CTEs of common metals.
In the section dedicated to material properties, Yoder discusses the properties of Invar. “It has a virtually invariable (hence the name) and low CTE over a limited temperature range (typically 40 to 100°F [4 to 38°C]). The CTE changes relatively rapidly outside this range.” As described in the MUVI grant proposal [12], the expected operating range for the MUVI instrument is 0°C - 40°C which would indicate Invar 36 would be a great material for use to bridge the CTE gap between the mirror and the flexures.

Design principles for space-based applications states that limiting contamination should be of critical importance. The standard method to install threaded fasteners is to use a lubrication to allow for consistent torque application to the fastener. However, many lubricants cannot be used due to their tendency to outgas and contaminate surfaces. Outgassing is the process of a material evaporating or sublimating while in the vacuum of space. The concern for optical instruments is that it will often condense on optical elements and reduce the efficiency of the optical system. In place of lubrication for fasteners, helicoils are often used as a mechanical method to provide consistent friction (k-factor) and therefore accurate torque during fastener installation. Additionally, any material that is evaluated shall be low outgassing, measured as Total Mass Loss (TML) or Collected Volatile Condensable Material (CVCM). For UV applications, the general guidelines is for TML to be less than 1% and CVCM to be less than 0.1% [13].

**Objectives and Thesis Scope**

The objectives of this thesis project are to design, build and test the mirror mounting system of MUVI. The mirror mounting system consists of flexures to isolate the mirrors and Invar bonding pads to bridge the different coefficients of thermal conductivity between the Fused Silica mirrors and Titanium flexures. Several flexure geometries are evaluated, and three front runner designs were selected for further evaluation in testing. These designs were for both the top and bottom mirrors and evaluated based on several criteria including their ability to isolate the mirror from vibration, manufacturability, cost and more.
The flexure blades were designed, analyzed, and optimized to isolate the mirror from the extreme vibration environment experienced during a rocket launch. During launch, there is an extreme amount of random vibration that excite all hardware and can cause severe damage if hardware is not rated for the loaded or otherwise isolated from said vibrational environments. The flexures’ requirement is to isolate the fragile mirrors from the vibration environments of launch.

NASA’s General Environmental Verification Standard [14] outlines the requirements to qualify the hardware to the extreme environments of launch through random vibration tests as well as sinusoidal sweep tests. The motivation for the sinusoidal sweep test is outlined in section 2.4.3 of GEVS:

“Sine sweep vibration tests are performed to qualify prototype/proto-flight hardware for the low frequency transient or sustained sine environments when they are present in flight, and to provide a workmanship test for all payload hardware which is exposed to such environments and normally does not respond significantly to the vibroacoustic environment, such as wiring harnesses and stowed appendages.”

While the sinusoidal sweep tests to experimentally measure the natural frequencies of the structure as well as tests for low frequency transients that were not captured in analysis. The sine sweep test is also utilized to measure change in the natural frequency before and after the assembly is subjected to the random vibration testing. The sine sweep test is performed immediately before and after the random vibration test cases. A change in the natural frequency after the random vibration test would indicate something failed or plastically deformed in test that alters the stiffness of the system. Essentially, verifying the natural frequency after the random test ensures that the assembly passed the test and nothing changed that would indicate it failed the qualification test.

The random vibration test cases are outlined in section 2.4.2 and are described, “As a screen for design and workmanship defects, components/units shall be subjected to a random vibration test along each of
three mutually perpendicular axes.” The random vibration test in GEVS calls out general use qualification levels for sub-assemblies as an amplitude spectral density at various frequencies from 20-2000Hz. The random vibration test is the main way of qualifying the design to flight like loading conditions. These tests are performed in the three mutually perpendicular axes. In the case of the mirror, this is the vertical axis, out of plane, and the two horizontal axes in place with the mirror.

Alongside the vibration testing performed on two shake tables, one configured for the vertical axis and the other for the two perpendicular horizontal axes, an autocollimator was used to measure the angular displacement or shift of the mirror through testing. The autocollimator is a tool to measure the optical alignment of the mirror. Prior to any vibration testing, the autocollimator is set to zero in both axes for the specific mirror and flexure assembly. Then after all testing outlined above, the mirror is placed under the autocollimator to quantify if the mirror experienced any permanent angular displacement. This, along with the pre and post-test sine sweep is used to ensure that the mirror and flexure assembly did not significant change or displace through testing and ensure the mirror would retain its required optical alignment. The goal in the testing phase is to see approximately 1 arc minute of angular tolerance or less.

In addition to the vibration testing of the assembly, epoxy bond coupon testing was performed to better understand and model the epoxy bond interface between the Invar bond pad and the mirror. Several coupons of overlapping aluminum plates were bonded together using the 3M 2216 Translucent epoxy and testing both in 3-point bend testing and tension to measure shear strength of the epoxy. The reason to experimentally test the epoxy used in bonding was to verify the shear strength of the epoxy would be sufficient for the designed bond area and expected loads of the Invar bonding pads. Additionally, these tests were performed to collect experimental data on how the epoxy responds under load to further refine and correlate a Finite Element Model of the bond which can then in turn be used to further refine the FEM of the mirror assembly.
Although there was extensive testing of the front runner designs, there was significant analysis of the flexures in the design phase of the project. Analytical approaches were utilized where possible, such as calculating the stress due to thermal effects and differences in the coefficients of thermal expansion between materials. For the more complex aspects, finite element analysis was used to perform a verity of analyses. From rough estimates of loading in a quasi-static environment to modal analysis of the systems natural frequencies and models of the random vibration environment that the flexures experience during vibration testing.
Chapter 2 – Development of an Optimum Flexure
This chapter outlines the design aspects of this project. Special focus is given to the flexure blades for the top and bottom mirrors, the modifications to existing parts, and specialized ground support equipment for assembly and testing. This chapter will discuss the overall final design as well as the process leading to the final design as well as the constraints and considerations that guided the design.

2.1 – Top Mirror Flexure Blades
A flexure blade is a compliant mechanism that is designed to flex under load. It is generally used when there is a sensitive component that must be mounted in such a way that it needs to be mechanically isolated from the rest of the system. In this application the flexures are thin pieces of titanium that will bend when the brittle fused silica mirror undergoes vibration. The flexure will take up the energy going into the system instead of passing it along to the mirror which could easily break due to its low strength and brittle nature. A flexure must be flexible but must have a high strength such that the deflections stay within the elastic range of the material and not surpass the yield point.

A requirement of the flexure is that it return to its original position after being subjected to vibrations. More details of this are discussed in the chapter on testing. The need for a high strength flexibility ratio alternatively measured as ratio of yield strength to elastic modulus makes titanium the clear choice of what material to use [10] Titanium has a ratio of 7.27 whereas other common metals like stainless steel is down at 4.39 and 6061 Aluminum is at 3.85 [15]. Additionally, the low density and therefore light weight properties of titanium make it a great choice for a space-based flexure design.

An additional design constraint for these flexures was that they must mount the existing 2” mirror selected for the optical path the light takes to the lens as well as mounting to the top door which had already been designed. The overall footprint of the top door is 3.67” x 3.46” as shown in the figure below.
During the design phase it was decided to develop three different flexure blade designs for both the top and bottom mirrors. During the design and analysis of the flexure blades, each different geometry was found to have its own benefits and drawbacks. It was decided that it would be best to design, analyze and test the top three leading designs. After extensive analysis and testing a frontrunner will be determined based on several criteria and a recommendation will be made as for which is the best design to implement in the instrument. Some of the criteria used to determine the overall best design include how well it attenuates vibration to the mirror, i.e. which blade is the best at isolating the mirror from stress induced by vibrational loading, the differences in natural frequencies between the designs and how they compare to the modes of the instrument frame as a whole, which design has the lowest manufacturing cost and if one design is easier to work with during assembly.
Figure 5. Top Mirror Curved Flexure Blade design.

Figure 6. Top Mirror Bent Flexure Blade design.
A number of flexure blade designs were considered including a straight blade, double curved blade and a zigzag blade but ultimately these three, the bent blade, the curved blade and the Z blade were found to be the most promising designs. The design calls for a three-point mounting scheme with three bonding pads located 120° of each other for a symmetric and properly constrained mount. Using a four-point mount was discussed but it would lead to an over-constrained boundary condition on the mirror. Three equally spaced mounting points around the perimeter of the mirror is ideal so as to not over constrain the mount.

The one drawback of using a titanium flexure blade with a fused silica mirror is the large disparity in coefficient of thermal expansion (CTE). The titanium used for the blades is grade 5, 6AL-4V alloy which has a CTE at room temperature of $8.6 \times 10^{-6} \text{ m/m/°C}$ [15] compared to the CTE of fused silica which is $0.54 \times 10^{-6} \text{ m/m/°C}$ [16]. This order of magnitude difference in CTE can produce major issues when the optic is subjected to temperature swings both in its operating environment and during assembly. A large difference in CTE is problematic because as the temperature increases the titanium would expand an order of magnitude amount more that the fused silica. This disparity in expansion would put a lot of stress into the bond between the mirror and the flexure blades. To mitigate this issue Invar with its relatively low CTE of $1.25 \times 10^{-6} \text{ m/m/°C}$ [17], was used as an intermediate between the flexure blades and the mirror. Invar is a nickel iron alloy that is very stable thermally and is used in a lot of applications when

**Figure 7.** Top Mirror "Z" Flexure Blade design.
there is a large CTE gap that needs to be bridged. The Invar bond piece serves as the bridge between the mirror and the flexure blades. Being a metal and much more ductile that the brittle fused silica mirror it can withstand the forces the titanium blade exerts during thermal expansion and contraction without passing it along to the mirror.

The Invar bond pad design.

The Invar piece will bond to the mirror with a strong structural adhesive and then the blades will mount to the invar with mechanical fasteners. The mirror requires an adhesive as the bonding agent as no mechanical fastener or other method of attachment would work. CubeSat specifications have very specific requirements on the types of adhesives that may be used. A Total Mass Loss (TML) of no more than 1% and a Collected Volatile Condensable Material (CVCM) of no more than 0.1% are required [18]. This is because when in the vacuum of space, many materials including adhesives will outgas, meaning that they give off gas into the vacuum. This can be very problematic with the sensitive electronics and optics both inside the MUVI instrument as well as the orbital deployer that holds the CubeSat during launch. 3M 2216 was chosen for its work life, strength and adherence to the TML and CVCM requirements. The 3M 2216 Translucent Epoxy has a TML of 0.77% and a CVCM of 0.04% [19], well within the CubeSat
specifications. Two 1.5mm holes on either side of the center of the invar pad are for a 17GA syringe tip to insert and apply the epoxy. 2216 epoxy is a viscous fluid while working with it, so a large tip opening is required to apply the epoxy to the surface. The epoxy has a 120-minute working life before it begins to set and takes a full 7 days to fully cure.

The mirror must be attached with an adhesive but there were more options available for attaching the flexure blades to the invar bond pads. There is flight heritage on the ICON mission [9] of using an adhesive to bond Titanium and Aluminum flexure blades to Invar parts. This option was discussed but due to the need for interoperability between the three different flexure blade designs it was determined a less permanent mounting method would be preferred. This naturally led to discussing threaded fasteners to easily swap between flexure blades while maintaining the same optic and Invar thermal bridge pieces.

One interesting feature of the Invar Bond Pad design is the shoulders at the top and bottom where the flexure blade attaches. This is to prevent the mirror rotating about the axis of the fasteners and moving out of alignment. The shoulders provide constraint in the rotational axis so that the mirror is in the exact location it is supposed to be in.

![Figure 9. Top Mirror with Z Flexure Blades, isometric and top views.](image-url)
2.2 – Bottom Mirror Flexure Blades
Much like the top mirror flexure blades, three designs were chosen to move forward into manufacturing for the bottom mirror flexure blades. Although there are many similarities in the design and overall look there were some unique challenges to the design of the bottom mirror flexure blades. The top mirror is a 2-inch diameter fused silica mirror whereas the bottom mirror is 1 inch in diameter. This smaller mirror resulted in a smaller area for bonding but due to the lower mass meant that the overall length of the flexure blades could be shorter. Another challenge of the design for the bottom mirror flexure blades was the volume constraints to fit the mirror around existing parts of the frame and the lens assembly.

Figure 10. Bottom Mirror Bent Blade Flexure Design

Figure 11. Bottom Mirror Curved Blade Flexure Design
To accommodate the geometric constraints of holding the mirror at the proper height and giving clearance to the lens assembly all three designs are at an angle to have a low-profile attachment point and rise to hold the mirror.

The bottom mirror is not as thick as the top mirror, so a redesign of the Invar bond pad was required to meet the geometric constraints of bonding to the bottom mirror. To maintain similarity in the design of the top and bottom Invar pads and to make the assembly process slightly easier the same size holes for epoxy access and the same size and depth of the hole for the fastener is used. Both invar pads will use a M2.5 4mm long vented screw to attach the flexure blades to the invar pads.
The unique design of the flexure blades and the requirement for high precision of the placement of the Invar bond pads called for a custom designed fixture to assist in the assembly and application of the epoxy for bonding. The Invar pads must be coplanar with the back of the mirrors as well as clocked at exactly 120° to ensure even distribution of the forces. In addition to those requirements the manufacturer of the 2216 epoxy recommends a 0.003” – 0.005” bond thickness [19]. The mirror must be held in place and a bond gap set before epoxy can be applied.
Figure 15. Top Mirror Assembly Fixture

The rim of the front of the mirror sits on a Delrin ring to center it in the fixture and to hold it at the proper height. Three blocks backed by spring plungers hold the mirror in place so that it doesn’t move. The three invar pads are held in place with holders that have slots to move back and forth. Mylar shims of 0.005” thickness are used to set the gap between the mirror and the invar pad. Once the bond gap is set the invar holders are screwed down and locked in place. Then the mylar shims can be removed and epoxy can be placed in the gap between the mirror and invar pads. A very similar setup and procedure is used for the bottom mirror assembly.

2.4 – Vibration Testing Mounting Fixture
Once all analysis of the flexures is complete, all designs will be tested through various vibration test procedures on the Shake Tables in the Cal Poly Mechanical Engineering Department Vibration Laboratory. The testing includes a pre-test visual alignment of the mirror with an autocollimator, sine sweep on a shake table to determine the natural frequency, random vibration tests at increasing intensity
as per the GEVS specification [14] and finally a post-test visual alignment to determine if the mirror and blades moved during testing. To test all six different flexure designs on the shake table a fixture was required to mount to the vibration shake table bolt pattern and have the proper hole pattern for all the different flexures. This fixture can mount the bottom mirror flexures on the inner ring of holes and the larger top mirror flexures on the outer ring of holes. Each flexure blade design, bent, curved and Z, are clocked 90° of each other to ensure no confusion as to which holes the flexures are supposed to thread into. This fixture can mount to either the horizontal or vertical shake table so that testing in all three principal axes, X, Y and Z can be conducted.

![Figure 16. Vibration Test Fixture](image)

![Figure 17. Horizontal Shake Table for X and Y axes (left) and Vertical Shake Table for Z axis (right)](image)
Chapter 3 – Analysis
This chapter discusses the analytical and numerical analysis that went into evaluating the flexure design and adhesive bonding for this project. Special focus is given to the flexure blades for the top and bottom mirrors and the bonding between the invar pad and mirrors. Finite Element Analysis (FEA) studies as well as hand calculations using analytical methods were performed to evaluate the design. This chapter discusses the analysis methodology and results. The following chapter will discuss the environmental testing that was conducted to validate the design and correlate the Finite Element Model (FEM) to the real-world results.

3.1 - Mirror Bonding Analysis
In order to determine the optimal bond area on the mirrors several factors must be considered. The three sources of stress on the bond are mechanical loading, both static and dynamic, shrinkage during curing and thermal expansion and contraction, both on ground and in orbit. Additionally, design for manufacturing and assembly was also considered when designing the overall shape of the Invar bonding pad. To better understand the mechanical and physical properties of the epoxy used several bond coupons were made. A finite element model of the bond coupons was made, and FEA results were compared with experimental data and the model was then correlated.

The three main loading conditions are the mirror mounting pads are thermal, vibration loading and shock loading. The thermal loading comes from the thermal expansion and contraction while in its operating environment as well. The 3M 2216 Translucent Epoxy used in the bonding the mirrors has a very slow cure time. At elevated temperatures the epoxy cures much faster but curing at the elevated temperatures is not advantageous for this application. The thermal window of the MUVI instrument is 0-40C. Curing outside of this range would mean that the bond cures at an elevated temperature and due to thermal expansion, the bond would be prestressed at the lower operating temperature since the bond would have been cured when the components are slightly larger at the elevated temperature. For this reason, it is
important that the epoxy cures at room temperature which is within the expected environmental temperature range. The vibration loading comes from when the instrument goes through launch and the shock loading comes from the door opening when the instrument is in operation. Due to the strength of the epoxy and the relative low mass of the mirrors it was determined the driving load case in determining the bond area was due to thermal stresses. This was expected because this was the root cause of failure in the original design of flexure blades in the MUVI phase II senior project. Additionally, simple shear in the bond is analyzed to ensure the bond meets both conditions.

Starting with the thermal analysis, several useful equations from Chapter 15 of Dr. Paul Yoder’s Opto-Mechanical Systems Design section 15.8 “Stress in Cemented and Bonded Optics due to Temperature Changes” were utilized to analyze the bond [20].

\[ S_S = \frac{(\alpha_M - \alpha_G)\Delta T S_e \tanh(\beta L)}{\beta t_e} \]  \hspace{1cm} (Eqn. 1)

\[ S_e = \frac{E_e}{2(1+\nu_e)} \]  \hspace{1cm} (Eqn. 2)

\[ \beta = \left[ \left( \frac{S_e}{t_e} \right) \left( \frac{1}{E_M t_M} + \frac{1}{E_G t_G} \right) \right]^{1/2} \]  \hspace{1cm} (Eqn. 3)

\( S_S \) is the shear stress in the bond, \( \alpha_M \) and \( \alpha_G \) are the CTEs of the metal and glass respectively, the “M” subscript is for the support which is generally metal and the “G” subscript for the optic which is generally glass, the “e” subscript is for the adhesive, and \( S_e \) is the shear modulus of the adhesive. Tanh is the hyperbolic tangent function; \( L \) the largest dimension (length, width, or diameter) of the bond; \( t_e \) is the thickness of the bond; \( E_M \) and \( E_G \) Young’s Modulus for the metal and glass, respectively; \( \nu_e \) the Poisson’s ratio for the adhesive; and \( t_M \) and \( t_G \) the thicknesses of the metal and glass components, respectively. The Beta, \( \beta \), function is the root of the two materials Elastics Modulus and temperature taking into account the shear modulus of the adhesive.
Using the first equation and the values found in equations 2 and 3, the shear stress in the bond can be determined from the thermal effects in the differences in CTE between the metal and the glass over the specified operating range of temperatures. The geometric properties of the joint that effect the shear stress are the thickness of each component, the adhesive, glass and metal and the largest dimension, L. The largest dimension can be either length, width, or diameter. With the material properties known for each of the materials used the shear stress due to thermal effects was obtained. See the table below that shows the results of the calculations. For comparison titanium and aluminum are included to show the difference the CTE of the metal bond pad makes in the shear stress. Titanium is the material of choice for the flexure blades and in the preliminary design phase 6061 Aluminum was considered as a flexure material too. Before the use of an Invar bonding pad was considered, a design for bonding the titanium directly to the mirror was evaluated and the table shows why it would be a bad decision based on the shear stress caused by thermal effects.

Table 1. Shear Stress due to thermal effects in the bond

<table>
<thead>
<tr>
<th>Material</th>
<th>CTE ([10^{-6}/°C])</th>
<th>Shear Stress [psi]</th>
<th>Factor of Safety</th>
</tr>
</thead>
<tbody>
<tr>
<td>Invar 36</td>
<td>1.25</td>
<td>141</td>
<td>12.06</td>
</tr>
<tr>
<td>Titanium 6Al-4V</td>
<td>8.6</td>
<td>1444</td>
<td>1.18</td>
</tr>
<tr>
<td>Aluminum 6061-T6</td>
<td>23.2</td>
<td>3564</td>
<td>0.48</td>
</tr>
</tbody>
</table>

The above table shows just how important it is to have a bonding pad that is close to the CTE of the mirror to reduce the thermal stresses in the bond. These values were calculated for the geometry of the top mirror bond pad and highlight why Invar was the chosen material. Given the same geometric properties of the bond the only thing changes between the materials is their Elastic Modulus, Poisson’s Ratio, and CTE. 3M, the manufacturer of the 2216 Translucent epoxy used in the bonding states that the allowable shear is 1700psi [19]. An aluminum bond pad would cause the bond to fail due to thermal effects alone as the calculated thermal stress in the bond would exceed that of the adhesive. Using titanium as the material
to bond with the mirror would have a positive safety factor at 1.18 but without taking into account any mechanical loading yet. To fully remove any concern of thermal effects, Invar 36 was analyzed due to the significantly lower CTE compared to most metals. When using Invar in the thermal calculations, due to the much lower difference in CTE between Invar and the mirror’s Fused Silica, the factor of safety came out to be 12.

Following the analytical calculation of the thermal stress in the bond, finite element analysis was used to numerically determine the shear stresses. Since the analytical method shows Invar as the clear winner it was the only material that was analyzed using FEA. After the completion of the thermal analysis of the bond, the mechanical loading of the bond was analyzed. The initial mechanical load case consisted of simple shear applied to the bond. A transverse shear force applied to the invar plug over the area of the bond is how to calculate the shear stresses in the bond. The Top Mirror Invar Pad has an area of 0.182in² (117.7mm²) based on the maximum allowable shear of 1700psi (11.7MPa) the bond can sustain a maximum shear force of 310 lbf (1380N) with a factor of safety of 2 which is both much lower that the dynamic loading when the flexure system is vibrating at its natural frequency and much lower than any forces exerted on the system during assembly.

The original mounting system designed by the second senior project group [4] had the challenges of a bonding area too small and bonded the aluminum flexure directly to the fused silica mirror. Both these shortcomings of the previous design of the bonding method have been addressed by the use of an invar bond pad. The Invar with its low CTE mitigated the thermal stresses that occur with the differences in CTE of the mirror and metal. The much larger bond area with the invar also corrects the issue of a stress concentration on the mirror from a small bond area. The larger area distributes any loads into the mirror over a larger area, ensuring the mirror will not fracture like with the original design.
3.2 – Precursor Analysis - Cantilever Beam FEA

Before moving on to analyzing the flexure blades a basic cantilever beam study was conducted using a finite element model and analytical equations to validate the analysis procedure in FEA. To evaluate the flexure design both static analysis and modal analysis of the flexure system was required. In order to ensure the proper methods were used in FEA, the cantilever beam was a simplified case to validate the analysis methodology.

Two cases were analyzed for validating the model. First a static load applied to the tip and deflection of the beam. The second case was determining the first natural frequency of the cantilever beam. The equation for deflection is [21]:

$$\delta = \frac{FL^3}{3EI}$$  \hspace{1cm} (Eqn. 4)

The deflection at the tip for a cantilever beam is a function of the force applied, F, the length of the beam, L, the elastic modulus of the material, E, and the moment of inertia, I. The analytical equation for the natural frequency of a cantilever beam is [22]:

$$f_n = \frac{3.52}{2\pi} \sqrt{\frac{EI}{wL^4}}$$  \hspace{1cm} (Eqn. 5)

The analytical equation for natural frequency is a function of E, I, F, and L like the equation for deflection but also has g the gravitational acceleration term and a constant out front and w which is the load per unit length, or in other words, the weight of the beam, normalized per meter. Dimensions of 400mm x 32mm x 6 mm and the material properties of 6061-T6 aluminum were used for the calculations. A unit load (1N) was applied to the end of the beam and a deflection of 0.5368mm was calculated. Given the dimensions and material properties a natural frequency of 30.66 Hz was found.
Following the analytical method was the numerical method. A cantilever beam was modeled both in SolidWorks Simulation package and Abaqus. Both FEA programs were used to compare solutions to each other as well as comparing to the analytical solution.

<table>
<thead>
<tr>
<th>Output</th>
<th>SolidWorks FEM</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement [mm]</td>
<td>0.5325</td>
<td>0.5368</td>
</tr>
<tr>
<td>First Natural Frequency [Hz]</td>
<td>30.88</td>
<td>30.66</td>
</tr>
</tbody>
</table>

3.3 - Flexure Blade Analysis
To refine the design of the flexure blade designs, extensive finite element analysis of the CAD models were performed. This section of the chapter goes into detail on the analysis, methodologies and results of the flexure blade assemblies. Initial sizing of the concept designs were performed using the simulation package in SolidWorks®, a commercially available 3D CAD and simulation software, followed by a refined model in Abaqus CAE®, an all-in-one software for creating finite element models, meshing, analyzing and post processing.

While performing initial sizing, quasi-static stress simulations for 100g load cases based on NASA JPL’s Mass Acceleration Curve (MAC) [23] were performed as well as modal analyses to determine the natural frequencies of each design. Determining natural frequencies was an important design requirement as the MUVI instrument structure has its first natural frequency around 800 Hz and it was important to avoid that frequency as this would lead to modal coupling. If the base structure and the mirror assembly were to have very similar natural frequencies then when the structure starts resonating, it would amplify the response of the mirror. The phenomenon is called modal coupling and is very important to avoid as the lead that the mirror sees can be amplified far beyond qualified limits.
Once each design had its rough shape and size refined using SolidWorks Simulation®, the 3D CAD models were saved as STEP files and imported to ABAQUS CAE®. Within ABAQUS, each model was built up and analyzed. In ABAQUS, the general workflow is to import all components, mate the assembly together and define constraints between parts. Then define properties of the model to include the material properties, boundary conditions and loads. Then the mesh of the model is defined. This is a highly critical step where different finite elements are selected, and the size of the elements are defined.

Once the model is fully created, a coarse mesh, meaning large elements, is defined and an initial analysis is run. The reason for doing the initial run with a large, coarse mesh is that it takes less time to solve and validates the model is setup properly. The finer the mesh, the more elements there are and as a result the longer time it takes to computationally solve the system of equations. Starting with a coarse mesh and refining it step by step is a method called mesh convergence. Mesh convergence is a step-by-step iterative approach to refine the mesh in a methodical way until the mesh is fine enough that a significant change in results is not seen.

Take the cantilever beam from the previous section as an example. This beam has a fixed connection at one end and a 1N point load at the other. This beam has dimensions of 6mm tall, 32mm wide and 400mm long. You might start out with elements that are 10 mm in size and get a stress distribution throughout the beam. Then you half the size of each element and see a change in the stress distribution. Keep repeating with halving the element size again and so on until the change in stress in the output of the analysis changes very little. This means that the mesh is now refined to the point that the results are accurate without putting in too many elements where the computation time would go up. Mesh convergence is an important part of creating a finite element model for two reasons, first to validate that the mesh is fine enough that the results you are seeing are accurate and second to a lesser extent, to ensure that the mesh isn’t so fine that extra computation is required beyond what is needed.
Table 3. Cantilever beam example of mesh convergence.

<table>
<thead>
<tr>
<th>Tip Deflection [mm]</th>
<th>10 mm element size</th>
<th>5 mm element size</th>
<th>2.5 mm element size</th>
<th>1.25 mm element size</th>
<th>Classic Beam Theory</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.5299</td>
<td>0.5318</td>
<td>0.5324</td>
<td>0.5325</td>
<td>0.5368</td>
</tr>
</tbody>
</table>

This mesh convergence study shows the importance of refining the mesh as large jumps were seen when reducing elements from 10 mm down to 5 mm and again down to 2.5 mm but the final step to 1.25 mm yielded almost no change in the tip deflection. Even with exponentially more elements in the finite element model, the tip deflection does not significantly change. As such, the mesh converged at the 2.5 mm element size, and further refinement yields no appreciably more accurate results.

Figure 18. Mesh convergence of cantilever beam. Coarse mesh (10mm element size) top left, medium mesh (5mm element size) top right, fine mesh (2.5 mm element size) bottom left and very fine mesh (1.25mm element size) bottom right.
Going back to the analysis of the flexures, once complete with refining the mesh and running mesh convergence studies on the flexure FEMs, analyses for quasi-static load cases were performed. These include the modal analysis to evaluate the natural frequency of the design, static stress to simulate the peak loading from vibrations and handling loads during assembly, and random vibration input analyses.

3.3.1 - Initial Finite Element Models
SolidWorks was the primary 3D CAD modeling software used for the design of all parts in this project. SolidWorks has a built-in simulation package used for general finite element analysis and was used for the rough sizing of all parts in the design. After the rough CAD models were complete, each design went through SolidWorks simulation to perform rough sizing. The initial load cases were 100g quasi-static loads based on the NASA Mass Acceleration Curve as well as 44.5N (10lbf) handling loads. The 44.5N handling load was used as a conservative estimate of what loads the flexures might see during assembly and integration of the mirror. The various loads of holding the flexure, mounting it to the mirror and mounting the mirror to the deployable cover are not well quantified so the blanket 44.5N handling load case was utilized to protect from any damage during assembly. These were rough load cases used to get the initial size of parts into the ballpark followed by more refinement later. Additionally, during the initial
load cases, the SolidWorks Modal Analysis tool was used to get a rough picture of the natural frequency of the design.

Although only flexure designs for the Top Mirror were manufactured and tested, as the overall goal of the project was to design the mounting mechanism for the full optical assembly and with that, the top mirror and lower mirror flexures were fully analyzed. The following sections show the finite element analysis of all six flexure designs.

3.3.2 – Top Mirror Flexure Analysis
The below sections outline the analysis for each of the three flexures for the top mirror. Each flexure design underwent the same iterative approach to determine the optimal shape for each design.

3.3.2.1 - Bent Flexure Blade – Top Mirror
Starting with the initial analysis of the bent flexure blade, the end condition where the flexure mounted to the mirror was a fixed point in space as well as the overall height of the flexure off the door but all other parameters were able to be modified. These include the blade thickness, the height of the blade, as well as the length and width. These could all be adjusted to fine-tune the frequency response and stress limits in the design. The below figures show the initial bent flexure blade stress analysis for a 44.5N load case as well as a modal analysis to analyze the modes of the system.

![Bent Blade mesh in SolidWorks.](image)

**Figure 20.** Bent Blade mesh in SolidWorks.
Using the native SolidWorks meshing tool, a fine mesh was selected for the finite element analysis. The mirror had a coarse mesh while the Bond Pad and Flexure had a fine mesh as these parts saw the most change when loaded. The below figure illustrates the load cases used for the SolidWorks Simulation. The base of each flexure is fixed in place to represent the end condition of being fastened down in place and the mirror sees a distributed load across the phase to represent a uniform acceleration load, in this case 44.5 N (10lbf) to test the load case due to handling loads.

![Figure 21. Bent Blade Z-axis load case](image)

The below plots show the stress results of the Bent Blade assembly when loaded in the three principal axes. The table below shows the peak stress for each load case and the corresponding factor of safety to yield strength. Notable about these load cases is that the highest stress seen in the length of the blade and not right at the corners. These were areas of concern as being high stress concentrations due to the 90° bend but through an iterative method, the corners were thickened and as a result the stress in those regions is lower that the length of the blades.
Figure 22. Bent Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center).

The above combination plot shows the high and low stress regions of the Bent Flexure blade in each of the three principal axes. The below table extracts the maximum stress experienced in each direction as well as the maximum deflection. The Max stress is compared to the tensile and ultimate strength of the material, in this case, Titanium 6AL-4V and the factor of safety to yield and ultimate is found. As shown below, the flexure assembly has a healthy 1.6 Factor of Safety to yield for the handling load cases.

<table>
<thead>
<tr>
<th>Load [N]</th>
<th>Direction</th>
<th>Max Stress [MPa]</th>
<th>Max Deformation [mm]</th>
<th>FS Yield</th>
<th>FS Ult</th>
</tr>
</thead>
<tbody>
<tr>
<td>44.5</td>
<td>Z</td>
<td>547.4</td>
<td>0.472</td>
<td>1.61</td>
<td>1.74</td>
</tr>
<tr>
<td>44.5</td>
<td>X</td>
<td>540.4</td>
<td>0.502</td>
<td>1.63</td>
<td>1.76</td>
</tr>
<tr>
<td>44.5</td>
<td>Y</td>
<td>551.1</td>
<td>0.473</td>
<td>1.60</td>
<td>1.72</td>
</tr>
</tbody>
</table>
Following the quasi-static stress analysis, the assembly underwent modal analysis. The modal analysis is a method to extract the natural frequencies of the structure and visualize the shape of each natural frequency. This is of interest for the purposes of the analysis as the natural frequency of the structure was one of the driving considerations in each design. The below figure shows the first three natural frequency mode shapes for the Bent Blade. The deformation is highly amplified to more clearly show the shape of the mode, although in real world testing, the assembly does not move that much. A measure of how much motion is expected for a given mode is the expected mass participation factor (EMPF). The EMPF is a measure of how much mass, on a scale on 0% to 100% is expected to participate at that mode. It is also a way to understand stress induced at each mode. The expected mass participation factor multiplied by the acceleration loading at that mode will give a quasi-static load that can be used for calculating the stress. This in turn can be used to calculate a quasi-static factor of safety when compared to the materials yield strength. The first natural frequency of a structure is where the most motion is seen and therefore has the highest mass participation factor, subsequent modes taper off from there. As seen here, the finite element analysis showed that there are five modes within the 20-2000Hz frequency range of interest but really only the first three modes have any significant mass participation, high modes, although present, had inconsequential mass participation.
Figure 23. Bent Blade Modal Analysis with high deformation for visualization. Horizontal axes (bottom) and vertical axis (top).

Table 5. Bent Blade Modal Analysis Results

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency [Hz]</th>
<th>Direction</th>
<th>EMPF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>196</td>
<td>Z</td>
<td>0.96</td>
</tr>
<tr>
<td>2</td>
<td>203</td>
<td>X</td>
<td>0.64</td>
</tr>
<tr>
<td>3</td>
<td>203</td>
<td>Y</td>
<td>0.64</td>
</tr>
<tr>
<td>4</td>
<td>350</td>
<td>X</td>
<td>0.02</td>
</tr>
<tr>
<td>5</td>
<td>350</td>
<td>Y</td>
<td>0.02</td>
</tr>
</tbody>
</table>

3.3.2.2 - Curved Flexure Blade – Top Mirror
For the Curved Flexure Blade, much of the same analysis was performed to fine-tune the geometry of the design. As with the Bent Flexure Blade, the overall height and connection point to the mirror were maintained so that the mirror is still in the same place with respect to the rest of the optical system but the blade thickness, length, radius of curvature and blade height were the main parameters to adjust. Throughout the analysis of the Curved Flexure Blade, the driving considerations were the factor of safety.
to yield on the 10lbf handling loads as well as making the system more compliant and reducing the natural frequency as this design was seen to be stiffer than the other two flexure designs.

Figure 24. Curved Blade Assembly mesh in SolidWorks.

Much like the Bent Flexure Blade, the mesh for the curved blade was defined to have the majority of the elements at the complex geometry of the flexure blade and bond pads and the mirror was a relatively coarse mesh. This finite element model had the same end condition constraints with the base of the flexures being fixed in place to represent a bolted joint and the mirror having a distributed load.
Figure 25. Bent Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center).

The Curved Blade finite element analysis shows that the design is strong enough to withstand the handling loads as a bounding case for loads that the flexure will see. The maximum stress seen in all axes is well bounded and a factor of safety to yield of 2 was calculated. The highest stress areas were right where the curved section of the flexure bends to attach to the Bonding Pad. This is expected as it is the thinnest portion of the blade and a sharp angle causes a stress concentration.

<table>
<thead>
<tr>
<th>Load [N]</th>
<th>Direction</th>
<th>Max Stress [MPa]</th>
<th>Max Deformation [mm]</th>
<th>FS Yield</th>
<th>FS Ult</th>
</tr>
</thead>
<tbody>
<tr>
<td>44.5</td>
<td>Z</td>
<td>432.6</td>
<td>0.112</td>
<td>2.03</td>
<td>2.20</td>
</tr>
<tr>
<td>44.5</td>
<td>X</td>
<td>372.5</td>
<td>0.144</td>
<td>2.36</td>
<td>2.55</td>
</tr>
<tr>
<td>44.5</td>
<td>Y</td>
<td>375.4</td>
<td>0.137</td>
<td>2.34</td>
<td>2.53</td>
</tr>
</tbody>
</table>
The results of the modal analysis of the finite element model shows that the Curved Blade has similar modes to the Bent Blade, although at higher frequencies. The Curved Blade has similar modal shapes with the out of plane, Z-axis being the lowest mode and the in-plane X and Y axis modes being tightly coupled near each other. The Curved Blade having higher natural frequencies indicates that the design is stiffer, as stiffness and natural frequency are inversely proportional. This is also seen in the quasi-static load cases where the Curved Blade sees lower deflection.
Table 7. Curved Blade Modal Analysis Results

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency [Hz]</th>
<th>Direction</th>
<th>EMPF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>395</td>
<td>Z</td>
<td>0.96</td>
</tr>
<tr>
<td>2</td>
<td>544</td>
<td>X</td>
<td>0.9</td>
</tr>
<tr>
<td>3</td>
<td>544</td>
<td>Y</td>
<td>0.9</td>
</tr>
<tr>
<td>4</td>
<td>730</td>
<td>X</td>
<td>0.03</td>
</tr>
<tr>
<td>5</td>
<td>730</td>
<td>Y</td>
<td>0.03</td>
</tr>
</tbody>
</table>

3.3.2.3 - Z Flexure Blade – Top Mirror
The Z Flexure Blade followed the same iterative approach to narrow down the shape and size of the design. The Z Flexure had the most parameters that could be changed; the overall length, the height of the blade portion and the two angles were all able to be changed. For simplicity the obtuse and acute angles were kept to whole numbers but there was a wide range of options to choose from. The overall footprint of the assembly had to stay within the design envelop of the door and mounting plate but this still left a lot of options for interaction.

![Figure 27. Z Blade Assembly mesh in SolidWorks.](image)

Just like the other two finite element models, the model for the Z Blade was built in the same way. Fixed constraints were placed on the feet of the flexures and a load across the whole mirror to represent loads
seen while handling and assembling the structure. The model was meshed with a fine mesh as seen in the above figure and then subjected to both quasi-static load cases and modal analysis.

**Figure 28.** Z Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center).

Comparing the results of the Z Blade to that of the Curved Blade and Bent Blade, it is evident that this design is less stiff and weaker than the other two designs. That is to be expected as the Z Blade design has the longest equivalent length, and that effect of the obtuse and acute angles mean that the flexure is more compliant as well as being subject to higher stress concentrations due to the sharper angles. The Z Blade had to be extensively modified to bring up the factor of safety to be above 1. The current design shows that the factor of safety is just a small amount above 1 at 1.15. The Z Blade is the thickest of the three
blades but due to the long length, stress concentrations form in the angles and at the attachment points which cause the stress to be just a bit under the yield strength of the material.

Table 8. Z Blade Handling/Assembly Static Stress (44.5N load)

<table>
<thead>
<tr>
<th>Load [N]</th>
<th>Direction</th>
<th>Max Stress [MPa]</th>
<th>Max Deformation [mm]</th>
<th>FS Yield</th>
<th>FS Ult</th>
</tr>
</thead>
<tbody>
<tr>
<td>44.5</td>
<td>Z</td>
<td>586.6</td>
<td>0.786</td>
<td>1.50</td>
<td>1.62</td>
</tr>
<tr>
<td>44.5</td>
<td>X</td>
<td>763.3</td>
<td>1.258</td>
<td>1.15</td>
<td>1.24</td>
</tr>
<tr>
<td>44.5</td>
<td>Y</td>
<td>767</td>
<td>1.347</td>
<td>1.15</td>
<td>1.24</td>
</tr>
</tbody>
</table>

Figure 29. Z Blade Modal Analysis with high deformation for visualization. Horizontal axes (top) and vertical axis (bottom).

As alluded to in the quasi-static finite element results, the Z blade is the least stiff and most compliant design. This was first shown in the high deflection under a static load and further confirmed through the modal analysis. The Z Blade has the lowest natural frequency by a large margin. The first mode of the Z blade at 123Hz is a third lower than the Bent Blade and nearly half of the Curved Blade. The low natural
frequency means that the flexure assembly is well below the natural frequency of the overall structure so no mode coupling should occur.

**Table 9. Z Blade Modal Analysis Results**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency [Hz]</th>
<th>Direction</th>
<th>EMPF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>123</td>
<td>X</td>
<td>0.86</td>
</tr>
<tr>
<td>2</td>
<td>123</td>
<td>Y</td>
<td>0.86</td>
</tr>
<tr>
<td>3</td>
<td>150</td>
<td>Z</td>
<td>0.96</td>
</tr>
<tr>
<td>4</td>
<td>264</td>
<td>Rotation about X</td>
<td>0.01</td>
</tr>
<tr>
<td>5</td>
<td>286</td>
<td>Rotation about Y</td>
<td>0.01</td>
</tr>
</tbody>
</table>

3.3.3 - Bottom Mirror Flexure Analysis
Moving onto the analysis for the flexures on the bottom mirror, much of the same analysis was performed. All flexures underwent modal analyses to verify the natural frequencies were within the appropriate ranges. The flexures also underwent the same 10lbf handling load test as they are small and delicate, so it was important to make sure they survive handling while keeping positive factors of safety.

3.3.3.1 - Bent Flexure – Bottom Mirror
Replicating the analysis procedure of the flexures on the top door, the bottom mirror Bent Blade flexures were analyzed for both stress due to handling loads and natural frequency using modal analysis. The same fixed constraints at the base of the flexures were used and the same 44.5N loading to the mirror. For the bottom mirror bent blade design, the parameters that were adjusted to dial in the analysis were the overall length, length of each bend segment, thickness and radius in each bend. The below figures and tables so the results of the finalized finite element model.
**Figure 30.** Bottom Mirror Bent Flexure Blade Assembly mesh in SolidWorks.

**Figure 31.** Bottom Mirror Bent Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center).
Starting with the stress analysis performed on the flexure assembly, this design holds up to the 44.5N force but with low safety factors. This design for the small 25.4mm bottom mirror is overall smaller so it is reasonable that there would be higher stress in the flexures. The 44.5N is a conservative load case to analyze the assembly under and is promising that the safety factors are positive.

**Table 10.** Bottom Mirror Bent Blade Handling/Assembly Static Stress (44.5N load)

<table>
<thead>
<tr>
<th>Load [N]</th>
<th>Direction</th>
<th>Max Stress [MPa]</th>
<th>Max Def [mm]</th>
<th>FS Yield</th>
<th>FS Ult</th>
</tr>
</thead>
<tbody>
<tr>
<td>44.5</td>
<td>Z</td>
<td>541.6</td>
<td>0.396</td>
<td>1.62</td>
<td>1.75</td>
</tr>
<tr>
<td>44.5</td>
<td>X</td>
<td>737.8</td>
<td>0.306</td>
<td>1.19</td>
<td>1.29</td>
</tr>
<tr>
<td>44.5</td>
<td>Y</td>
<td>768.9</td>
<td>0.302</td>
<td>1.14</td>
<td>1.24</td>
</tr>
</tbody>
</table>

**Figure 32.** Bottom Mirror Bent Blade Modal Analysis with high deformation for visualization. Mode 1, normal to mirror (top left), rotation about mirror plane mode 2 (top right) and mode 3 (bottom center).
Following the stress analysis, the modal analysis was performed. These mirrors had much different modal shapes than that of the top mirror flexure designs but still well within the allowable bounds for the natural frequencies. The common mode shapes of the top mirror flexures were translational with rotational modes only being present at much higher frequencies. Comparatively, the Bottom Mirror Bent Blade shows that the first mode is translational in nature, moving out of plane, normal to the mirror, but all the other modes are rotations about axes on the mirror. The below table shows the first five modes as well as the Expected Mass Participation Factor or EMPF which is a measure of how much of the assembly is excited at each mode. For the first mode, 60% of the assembly is excited at that frequency.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency [Hz]</th>
<th>Direction</th>
<th>EMPF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>429</td>
<td>Normal (X,Z)</td>
<td>0.60</td>
</tr>
<tr>
<td>2</td>
<td>463</td>
<td>Rotation about X</td>
<td>0.58</td>
</tr>
<tr>
<td>3</td>
<td>463</td>
<td>Rotation about Y</td>
<td>0.58</td>
</tr>
<tr>
<td>4</td>
<td>796</td>
<td>Rotation about X</td>
<td>0.15</td>
</tr>
<tr>
<td>5</td>
<td>796</td>
<td>Rotation about Y</td>
<td>0.15</td>
</tr>
</tbody>
</table>

3.3.3.2 - Curved Flexure – Bottom Mirror
The curved blade underwent the same analyses, and the finite element was setup in the same fashion. The unique design of the curved flexure allowed for a lot of parameters to be edited in the iterative analysis process. The bottom mirror’s curved flexure design, unlike the top mirror flexure, has two radii of curvature to adjust on top of the thickness and width of the flexure. By modifying the curvatures separately, more refinement options were available. The below section highlights the finalized design of the Bottom Mirror Curved Flexure design.
Figure 33. Bottom Mirror Curved Flexure Blade Assembly mesh in SolidWorks.

Figure 34. Bottom Mirror Curved Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center).
Starting with the stress analysis for the 44.5N load case, the curved design, much like the Bent Blade flexure, has quite low safety factors compared to the top mirror, 1.18 compared to 2.03 on the top mirror curved blade. That being said, the Bottom Mirror Curved flexure still holds positive margin and passes the handling load analysis.

Table 12. Bottom Mirror Curved Blade Handling/Assembly Static Stress (44.5N load)

<table>
<thead>
<tr>
<th>Load [N]</th>
<th>Direction</th>
<th>Max Stress [Mpa]</th>
<th>Max Def [mm]</th>
<th>FS Yield</th>
<th>FS Ult</th>
</tr>
</thead>
<tbody>
<tr>
<td>44.5</td>
<td>Z</td>
<td>722.6</td>
<td>0.372</td>
<td>1.22</td>
<td>1.31</td>
</tr>
<tr>
<td>44.5</td>
<td>X</td>
<td>695.4</td>
<td>0.251</td>
<td>1.27</td>
<td>1.37</td>
</tr>
<tr>
<td>44.5</td>
<td>Y</td>
<td>745.8</td>
<td>0.257</td>
<td>1.18</td>
<td>1.27</td>
</tr>
</tbody>
</table>

Figure 35. Bottom Mirror Curved Blade Modal Analysis with high deformation for visualization. Mode 1, normal to mirror (bottom center), rotation about mirror plane mode 2 (top right) and mode 3 (top left).
The Curved Flexure exhibits similar behavior to the Bent Blade Flexure on the bottom mirror. The first mode is translational moving perpendicular to the plane of the mirror but the following modes are rotational. The modal analysis also found that the Curved flexure is stiffer than the bent blade with the first natural frequency being at 458 Hz compared to 429 Hz of the Bent Blade.

**Table 13. Bottom Mirror Curved Blade Modal Analysis Results**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency [Hz]</th>
<th>Direction</th>
<th>EMPF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>458</td>
<td>Normal (X,Z)</td>
<td>0.60</td>
</tr>
<tr>
<td>2</td>
<td>534</td>
<td>Rotation about X</td>
<td>0.33</td>
</tr>
<tr>
<td>3</td>
<td>534</td>
<td>Rotation about Y</td>
<td>0.47</td>
</tr>
<tr>
<td>4</td>
<td>1220</td>
<td>Rotation about X</td>
<td>0.28</td>
</tr>
<tr>
<td>5</td>
<td>1220</td>
<td>Rotation about Y</td>
<td>0.41</td>
</tr>
</tbody>
</table>

**3.3.3.3 - Z Flexure – Bottom Mirror**

To close out the analysis on the bottom mirror flexures, the Z Blade was analyzed, looking at stress due to a 44.5N load and the modal analysis. The Z blade was the most unique design of all having the same thickness and width parameters to adjust but also the angles of the interior and exterior bends. This led to an iterative approach where multiple studies were conducted until positive safety factors on loads were seen and acceptable natural frequencies were determined. Just like all other models, the base of the three flexures were fixed in place and the mesh was built up as seen in the below figure.
Figure 36. Bottom Mirror Z Flexure Blade Assembly mesh in SolidWorks.

Figure 37. Bottom Mirror Z Blade handling load stress plots in X-axis (top left), Y-axis (top right) and Z-axis (bottom center).
Unlike the Z blade of the top mirror, the Z blade on the bottom mirror proved to be stronger in the direction out of plane of the mirror but the other directions were very similar. The bounding safety factor in the X direction at 1.15 is quite similar to that of the top mirror Z blade and the other bottom mirror flexure. Although the stress is rather high, the design still holds positive margin at the 10 lbf load in all directions.

**Table 14.** Bottom Mirror Z Blade Handling/Assembly Static Stress (44.5N load)

<table>
<thead>
<tr>
<th>Load [N]</th>
<th>Direction</th>
<th>Max Stress [MPa]</th>
<th>Max Def [mm]</th>
<th>FS Yield</th>
<th>FS Ult</th>
</tr>
</thead>
<tbody>
<tr>
<td>44.5</td>
<td>Z</td>
<td>446.0</td>
<td>0.406</td>
<td>1.97</td>
<td>2.13</td>
</tr>
<tr>
<td>44.5</td>
<td>X</td>
<td>766</td>
<td>0.586</td>
<td>1.15</td>
<td>1.24</td>
</tr>
<tr>
<td>44.5</td>
<td>Y</td>
<td>752</td>
<td>0.592</td>
<td>1.17</td>
<td>1.26</td>
</tr>
</tbody>
</table>

**Figure 38.** Bottom Mirror Z Blade Modal Analysis with high deformation for visualization. Mode 1, normal to mirror (bottom center), rotation about mirror plane mode 2 (top right) and mode 3 (top left).

Finally, the Z blade underwent the modal analysis and found the natural frequencies of the flexure assembly. The most interesting finding here was that the Z blade proves to be the most right as shown by
the highest natural frequency of the group. Additionally, the Z Blade has its first two modes as the rotational modes and the translational mode is at a higher frequency. This is quite different than that of all the other flexure designs. This seems to be due to the angles of the “Z” in relation to the mirror. Since the flexures on the bottom mirror come up at an angle from the mounting plate, the “Z” portion of the blade is on a different plane than the mirror, different from the Top Mirror design. This gives the mirror more rigidity in rotation and therefore the natural frequency is higher.

Table 15. Bottom Mirror Z Blade Modal Analysis Results

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency [Hz]</th>
<th>Direction</th>
<th>EMPF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>555</td>
<td>Rotation about X</td>
<td>0.54</td>
</tr>
<tr>
<td>2</td>
<td>556</td>
<td>Rotation about Y</td>
<td>0.51</td>
</tr>
<tr>
<td>3</td>
<td>616</td>
<td>Normal (X,Z)</td>
<td>0.61</td>
</tr>
<tr>
<td>4</td>
<td>1160</td>
<td>Rotation about X</td>
<td>0.004</td>
</tr>
<tr>
<td>5</td>
<td>1220</td>
<td>Rotation about Y</td>
<td>0.003</td>
</tr>
</tbody>
</table>

3.3.4 - Further developing the Finite Element Models
With all the model geometry refined within the initial SolidWorks Simulation software, the next step was to move all analysis into Abaqus CAE which has much more refined controls for more accurate finite element analysis. This section will outline the general steps taken to setup the model and run the analysis in Abaqus CAE.

The first step in building the model is to import all the constituent parts and create the assembly. To do this, all files were converted from their native SolidWorks file type to a STEP file and imported into Abaqus CAE. From there the part constraints were defined to start to get the model to assemble together. One each of the parts were defined in relation to each other, as shown in the below figure, various properties of the model could be defined. Those include setting the global units such as mm, kg, N etc for these models and defining material properties. The below table shows the material properties used in the model to define each part’s properties.
Table 16. Material properties used for mirror-flexure FEM

<table>
<thead>
<tr>
<th>Material</th>
<th>Mass Density [kg/m³]</th>
<th>Young’s Modulus [GPa]</th>
<th>Poisson’s Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Titanium 6AL-4V</td>
<td>4429</td>
<td>104.8</td>
<td>0.31</td>
</tr>
<tr>
<td>Invar</td>
<td>8055</td>
<td>141.3</td>
<td>0.26</td>
</tr>
<tr>
<td>Fused Silica</td>
<td>2200</td>
<td>70.3</td>
<td>0.17</td>
</tr>
</tbody>
</table>

With the model built and the properties define, the next step is to define the constraints within the system. For this case, just like the SolidWorks simulation, the base of each flexure is fixed in place as that is a bolted joint and for simplicity represents a fixed point.

The next step in creating the model is to define the mesh. The mesh definition process is complex and an important step in the process of setting up the model. For the flexure and bonding pads, a tetrahedral 4 node C3D4 element was selected to model complex the three-dimensional geometry. For the mirror as it is a simpler shape and has less complexity, the isoparametric C3D8 element was selected. The C3D8 is a three-dimensional 8 node brick element. This element type is essentially a cube with nodes at each corner.
Figure 40. Illustrative example of a C3D8 element type [24]

See the below figure for a reference of the mesh. When meshing, there is the option to define the mesh at a part specific level. This level of control is a powerful tool as the flexures themselves have a lot of complex geometry and benefit from a lot more elements whereas the mirror is a relatively simple shape with less complex interactions and can get away with few elements.

Figure 41. Top Mirror Bent Blade mesh
With the model generated, the first test was a simple load case. In the above figure, a 15g load, corresponding to about 1N of force was applied to the mirror to observe how the assembly reacted.

Following the stress analysis which closely matches that of the SolidWorks FEM, the more complex operations that Abaqus has can be utilized. This includes the modal analysis to determine the natural frequencies and Random Response simulations to simulate the assembly when it goes through the random vibration testing.

Setting up the Random Response study builds off the already defined model and inputs the NASA GEVS random response curve in as well as an estimated 1% damping factor. The output of the study gives a plot that shows peak acceleration response at its natural frequency when the random vibration is inputted. The below plot shows the response of the Bent Flexure Blade when the random vibration in inputted in the Z direction. The units are mm/s\(^2\) so this spike corresponds to a roughly 150g input.

![Figure 42. Bent Blade Flexure random response study](image)

The Random Response Study is to verify that the flexure designs can withstand the random vibration environment as part of the qualification testing. This plot above shows that the flexures could see up to
150g acceleration. Based on a 62g mass of the assembly and the 44.5N input force, the flexures have been
designed to withstand a quasi-static 700g acceleration so are well bounded by this 150g input seen in the
finite element analysis.

3.4 – Glue Test Specimen FEM
The final analysis is of the adhesive modelling and testing to characterize the 3M 2216 Epoxy used for
bonding the Bonding Pads to the mirrors. The below table lists the material properties of the aluminum
bars and the epoxy adhesive used for the testing. Just like building the flexure models in Abaqus for stress
and vibration testing, the adhesive models were built in the same way. First by defining the constituent
parts, then by assigning interactions, constraints and load cases, then by meshing the parts and finally
running the study.

![Figure 43. Three-Point Bend Adhesive Test Specimen](imageURL)
Figure 44. Tension Adhesive Test Specimen

<table>
<thead>
<tr>
<th>Material</th>
<th>Mass Density [kg/m$^3$]</th>
<th>Young’s Modulus [GPa]</th>
<th>Poisson’s Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>6061-T6 Aluminum</td>
<td>2700</td>
<td>69</td>
<td>0.33</td>
</tr>
<tr>
<td>3M 2216 Epoxy [2]</td>
<td>1356</td>
<td>1.027</td>
<td>0.46</td>
</tr>
</tbody>
</table>

Following the testing of the epoxy bond coupons the finite element model that was previously developed was further refined using the experimental data. FEMs for both the bending and tension configurations were simultaneously correlated to the test data. The two measured values from the test were load and displacement, therefore these are the values used in the refinement of the model. For the bend test FEM a load of 250N was applied. This corresponds to an experimental displacement of 2mm. The parameters that were adjusted were the Elastic Modulus of the epoxy and the Poisson’s Ratio. Additionally, slight modifications of the aluminum’s thickness within the manufactures specified tolerance.
For the tension test FEM, a load of 2500N was used which corresponds to a displacement of 0.5mm. The same parameters as with the bend test FEM were modified.

It was challenging to get results of the FEM to line up with the experimental test results. The main reason for this was modeling such a thin material. The epoxy is only 0.005” thick compared to the 0.125”
thickness of the aluminum bars. The finite element model for the three-point bend was able to be correlated and a close displacement value was obtained when compared to the test results. The finite element model results for the tension case were an order of magnitude off when compared to the experimental results this is mainly due to the shear in the very thin epoxy bond causing high levels of deformation of the finite elements and not accurately capturing the true deformation.
Chapter 4 – Testing

4.1 - Testing Overview
The testing for this project consists of material characterization of the epoxy bonding used for bonding the mirror and Invar, vibration testing of the mirror-flexure system and post vibration test optical alignment. The material characterization consists of testing epoxy in tension and bending and correlating results to a FEM for use in modeling the mirror-flexure system. Vibration testing consists of Sine signature test to determine the natural frequency and random vibration input following the specification of GEVS section 2.4 on mechanical testing of instrument hardware. These tests would be done on X and Y horizontal axes as well on the z-axis, vertically for all six flexure geometries. Following each test an autocollimator is used to determine if there is any shift to the mirror’s orientation.

4.2 - Epoxy Coupon Testing
To better understand the material properties and failure modes of the adhesive bond test coupons were made to undergo bend and tensile tests. Ten specimens were made for bend testing and ten for tensile testing. The same 3M 2216 Translucent Epoxy was used and followed the same preparation procedures as used in the Invar-Mirror bond. The goal of these tests were to collect experimental data on how the epoxy responds under load to further refine and correlate a Finite Element Model of the bond which can then in turn be used to further refine the FEM of the mirror assembly. Test data gives load and displacement data which can be used in the refinement of parameters such as Elastic Modulus, E, Shear Modulus, G, and Poisson’s ratio, ν.

To start a couple specimens were made using aluminum pieces cut from bar stock and a test mix of the 2216 epoxy to practice the procedures of surface prep, mixing and applying the epoxy. One practice bend test specimen and one practice tension test specimen were made to check any problems in the procedure before making the twenty total specimens. Surface prep consists of cleaning the metal with isopropyl alcohol, removing the top oxide layer by scrubbing with Scratch-Brite and then cleaning again to remove any debris from the scuffing process. The epoxy is mixed in a 1:1 ratio of the base and accelerator and has
a 2-hour work life so while important to have everything prepped before mixing it is ok to take time to make sure application is done carefully. To replicate the bond line used in the Invar-Mirror bond as well as following the manufacturers recommended specifications for bond strength, a .005” bond line was used [1]. To obtain a consistent and accurate gap a sheet of PET plastic with .005” thickness was used to set the gap.

**Figure 46.** Schematic of bonding setup to ensure consistent bond gap.

The figure above shows a schematic of using another piece of metal with the same thickness and then a .005” piece of plastic to set the gap between the top and bottom pieces of aluminum. There are two concerns with the method, first being a tolerance stack-up between the aluminum and the plastic shim. Both are off the shelf parts and have small tolerances, so it was deemed to be close enough for the goal of these tests. The second concern was that there would be some flexibility in the overhung aluminum piece, and it would deflect. This too was determined to be negligible and would not have a significant effect on the thickness of the bond. With the process and procedure of making the epoxy tests coupons dialed in, the twenty test specimens were then constructed. The aluminum used is 6” x 1.5” x 1/8” 6061-T6. This was used because it is a common size that could be ordered in the 6” length instead of requiring extra processing in house before assembling and 6061 was chosen because it is one of the most common aluminum alloys available making procurement easier.
After allowing the proper amount of cure time according to the manufacturer’s specifications the coupons were ready for testing. The two testing standards used were ASTM D1184 “Standard Test Method for Flexural Strength of Adhesive Bonded Laminated Assemblies” [25] and ASTM D897 “Standard Test Method for Tensile Properties of Adhesive Bonds” [26]. These standards were used in setting up the material testing machine to have the correct parameters, most importantly the rate of extension. For bending the rate of extension is governed by the equation [25]:

\[ N = \frac{ZW^2}{6d} \]
Where $N$ is the rate of extension in mm/min, $Z$ is the unit rate of fiber strain which is a constant for this test and a value of 0.01 mm/mm/min was used, $L$ is the overall length of the bending area, for this test it is 152.4 mm (6 in.) and $d$ is depth of the beam. Based on the geometry of the specimen this is 1.03 mm/min and 1 mm/min was used when setting up the test parameters. According to ASTM D897 tension tests should not exceed 1.3 mm/min so the same value of 1 mm/min was used in tension as well. All testing was conducted on the same day on the same machine. The AMETEK LD50 in Cal Poly’s Mechanical Engineering Composites Laboratory. The ambient temperature was 26°C in the room.

![Figure 49. AMETEK LD50 Tensile Testing machine](image)

For the bend testing the top roller pressing down was placed at the midpoint of the 1.5” square bond. The roller was also at the midpoint of the 3-point span at 3” from each of the bottom rollers. This was kept consistent for all tests. The figure below shows the test setup of the specimen in the machine.

![Figure 50. Three-Point Bend Test setup](image)
All ten specimens failed uniformly, at roughly 450N they would fly apart. The failure mode was an adhesion failure as described in the ASTM testing specification. “Adhesion failure refers to the lack of adhering to metals being fastened” [25]. The bonds all failed where one piece of metal had a majority of the epoxy remain attached and the other had less, it was not a 50/50 distribution. Additionally, it was not a cohesive failure mode where “the failure has occurred in the adhesive itself” [25].

The Bend tests results followed a mostly linear trend up to the 2.5mm displacement mark when the slope of the force vs displacement was no longer a straight line. This indicates that this is where plastic yielding was occurring, and the bonding was beginning to fail. All ten specimens were close to each other, followed the same trend and had similar ultimate failure points. The only specimen somewhat outside of the others was specimen 3 but even still the standard deviation on the maximum load was 5% and under 4% for the displacement.
Figure 52. Force vs Displacement results for the bend test of epoxy bond coupons.

The plot of the average bend test results was created by taking the average of all specimen values at each displacement value. The average plot shows the trend of linear elastic response up to about 2.5mm and the plastic yielding occurring after that point. This average plot is used in refining the FEM of the bond by finding an associated displacement occurring for a given load.
Figure 53. Average Force vs Displacement results of the bend tests for epoxy bond coupon testing

In addition to analyzing the overall trend of the data, the failure point of each specimen was examined. Having consistency in the failure point indicates a good process for mixing the epoxy and applying it to the metal. If there was a very wide range of loads that caused the bond to fail that would tell us that the method of bonding was not consistent and should be addressed. Since the standard deviation of the data set is relatively small for the rigor of these tests it indicates that the bonding process is adequate and meets the requirements of this application.
Table 18. Failure point of bend test specimens for epoxy bond coupon testing

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Max Load [N]</th>
<th>Max Displacement [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>424.3</td>
<td>4.71</td>
</tr>
<tr>
<td>2</td>
<td>452.5</td>
<td>4.95</td>
</tr>
<tr>
<td>3</td>
<td>404.8</td>
<td>4.83</td>
</tr>
<tr>
<td>4</td>
<td>448.6</td>
<td>4.93</td>
</tr>
<tr>
<td>5</td>
<td>482.6</td>
<td>5.19</td>
</tr>
<tr>
<td>6</td>
<td>442.3</td>
<td>4.80</td>
</tr>
<tr>
<td>7</td>
<td>458.2</td>
<td>5.03</td>
</tr>
<tr>
<td>8</td>
<td>479.3</td>
<td>5.27</td>
</tr>
<tr>
<td>9</td>
<td>454.0</td>
<td>5.00</td>
</tr>
<tr>
<td>10</td>
<td>449.4</td>
<td>4.86</td>
</tr>
<tr>
<td>Mean</td>
<td>449.6</td>
<td>4.96</td>
</tr>
<tr>
<td>Standard Deviation</td>
<td>23.0</td>
<td>0.18</td>
</tr>
</tbody>
</table>

After testing the bending specimens, the tensile test specimens were tested. All tests were on the same day and machine to ensure consistent test conditions. The tensile test specimens are two top pieces bonded to a single bottom piece. This was done so that the uniaxial load condition was achieved and no bending in the bond would occur. If the single overlap orientation, as used for the bend test specimen, for tensile testing the misalignment of the top and bottom pieces would cause a bending moment to be induced. 1.5” of the specimen’s length was placed into each of the jaws so the overall length of the of the test section was 7.5” for each specimen. For all tests the two aluminum pieces with a spacer were clamped into the top jaw and the single aluminum piece was clamped into the bottom jaw.
All ten specimens had a similar response to loading and had the same failure mode. The exhibited linear behavior as the displacement increased the load increased proportionally. All but one had nearly the same slope. As shown in the figure below, specimen number 10 had a shallower slope. This is most likely due to improper bonding resulting in a joint with different properties. While all but one had the same slope, there was a bit larger range in the max load and displacement that each specimen experienced. Specimens 2, 5 and 8 look to be stronger than the rest.

The bonds failed in such a way that one of the bonds would fail and almost instantaneously the second bond would fail too stopping the test. The bonds failed but unlike the bend test, the metal pieces weren’t fully separated. There was epoxy on both pieces still holding them loosely together. This indicates a cohesive failure mode where the adhesive itself was what failed, not the bonding interaction with the metal.
Figure 56. Tensile test results of epoxy bond coupon testing

The figure below shows the average curve of all ten specimens. With a large enough sample size, it is clear that the bonded joint exhibits linear elastic behavior up to 2.7mm of displacement where sudden failure occurs. This plot will be used for the correlation of the FEM that was built to model the epoxy bond.

Figure 57. Average Displacement vs Load plot for Tensile testing of epoxy bond coupons

While the bend test results had fairly close failure points, the tensile test saw a somewhat larger spread of the data. The table below shows the failure load and associated displacement for each specimen as well as
the average and standard deviation of the sample set. The bend test had a standard deviation of around 5% of the average whereas with the tensile test the standard deviation is closer to 10%, not as desirable.

Table 19. Failure point of tensile test specimens for epoxy bond coupon testing

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Max Load [N]</th>
<th>Max Displacement [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10890</td>
<td>2.79</td>
</tr>
<tr>
<td>2</td>
<td>12708</td>
<td>3.10</td>
</tr>
<tr>
<td>3</td>
<td>10894</td>
<td>2.98</td>
</tr>
<tr>
<td>4</td>
<td>8671</td>
<td>2.24</td>
</tr>
<tr>
<td>5</td>
<td>11098</td>
<td>2.74</td>
</tr>
<tr>
<td>6</td>
<td>10694</td>
<td>2.65</td>
</tr>
<tr>
<td>7</td>
<td>10190</td>
<td>2.73</td>
</tr>
<tr>
<td>8</td>
<td>11899</td>
<td>3.06</td>
</tr>
<tr>
<td>9</td>
<td>10605</td>
<td>2.76</td>
</tr>
<tr>
<td>10</td>
<td>9582</td>
<td>3.06</td>
</tr>
<tr>
<td>Mean</td>
<td>10723</td>
<td>2.81</td>
</tr>
<tr>
<td>Standard Deviation</td>
<td>1122</td>
<td>0.26</td>
</tr>
</tbody>
</table>

One explanation for the larger spread in the peak loading the specimens could sustain was the added complexity in assembling and bonding the three pieces of metal compared to just two pieces with the bend test coupons. For the tensile test coupons, two metal pieces were bonded and then 24 hours later once the epoxy had reached its working lifetime and could be handled the third piece of metal could be bonded. Since two bonds are required for the tensile coupons, this doubles the chance of getting a bad bond and makes it more likely to have a larger range of results.

4.3 - Invar Plug to Mirror Bonding
Prior to any vibration testing, the mirror’s mounting system must be assembled. The mirror bonding assembly fixture, as described in the design chapter, was used to bond two pairs of mirrors. The fixtures were assembled, and the mirror and invar were prepped for bonding. Similar to the surface prep of the epoxy bond specimens, the mirror and invar were scuffed up with Scotch-Brite and then cleaned with isopropyl alcohol to remove any contaminates from the surface, ensuring a good bond. Following surface
preparation, the mirror was placed face down and the three spring loaded blocks were brought in to constrain movement of the mirror. With the mirror locked in place the invar pads were tightened down to ensure planarity with the back surface of the mirror. This was done by using an off the shelf 1.1” diameter flat glass lens Thor Labs to act as a flat plate. With the loose invar pads aligned with the mirror they were tightened down locking them in place. Then 0.005” mylar shims were used and the invar holder with its slots was brought in to set the proper bond gap as specified by the manufacturer. Tightening the invar holder into the base completes the setup process. The shims were then removed, and the mirror was ready for bonding.

![Bottom mirror (left) and Top mirror (right) bonding fixtures.](image)

**Figure 58.** Bottom mirror (left) and Top mirror (right) bonding fixtures.

The epoxy was mixed and applied to the gap between the mirror and invar using a syringe that fit in the holes in the invar piece. The 2216 Translucent epoxy is fairly thick and viscous requiring a sufficiently large syringe to be able to let the epoxy flow into the bond gap. A 17 Ga syringe tip was used and seemed to work well for the viscosity of the epoxy. The biggest challenge with applying the epoxy was that the two fill holes in the invar are a bit too far apart to allow for even application of the epoxy in the direct center of the bonding region. In order to ensure full coverage of epoxy in the bond gap it was necessary to overfill and cause the epoxy to spill out the sides. This is less than ideal but necessary to ensure a strong and consistent bond across the whole area.
Figure 59. Bottom mirror with overfilled epoxy bond.

As shown in the figure above, the epoxy seeped out the sides of the invar piece. This was a big challenge for the bottom mirror due to how small all the parts are and how close everything is together. Any small amount of epoxy escaping had a high chance of creating an undesirable bond with other parts. During the first bonding attempt for the lower mirror, epoxy dripped down and got on the Delrin ring causing the front rim of the mirror and the Delrin to bond together. Although this mirror was still useable in vibration testing this would be unacceptable for a mirror going on the MUVI instrument. In the figure below the rim of the mirror and the Delrin alignment ring has visible epoxy residue.
To prevent this from happening again, a donut shape with a 1-inch diameter was cut from the plastic used as shims and then used as a seal between the mirror and Delrin, beneath the invar and above the Delrin. On the second attempt of bonding invar to the mirror the donut was used, and the results were much better than in the first bonding attempt. In the figure below look for the thin piece of plastic going around the mirror beneath the spring-loaded pieces and invar parts.

**Figure 60.** Bottom mirror and Delrin ring after first bond attempt.

**Figure 61.** Bottom mirror bonding fixture on the second attempt with plastic seal
The second time bonding invar to a bottom mirror had a much better outcome than the first time. The figure below shows the mirror and Delrin. There is no longer epoxy on the rim or front face of the mirror nor is there any epoxy on the Delrin ring. It is important to note that this was not an issue with the larger top mirror. This is primarily due to the larger distances between parts as well as the fill holes in the top invar parts being closer to equidistant from each other and the edges whereas the bottom mirror invar parts have fill holes much closer to the edges to accommodate the M2.5 fastener required for the flexures.

![Image of mirror and Delrin](image)

**Figure 62.** Second bottom mirror bonded having used plastic seal.

Overall, the mirror bonding went well but there were some challenges in the process that could be refined in a future iteration. As mentioned previously the bottom invar parts could greatly benefit from a redesign. Placing the two fill holes at one third and two thirds of the length of the invar would ensure balanced application of the epoxy and not require the overfilling that is currently needed to achieve a proper bond in the center of the bond area. The plastic donut worked well as a method of solving the problem presented by the current design but if a future iteration were to be made then it is recommended to make a design modification with epoxy bonding in mind.
The point of comparison for these mirror bonds is that of the flexure design of the Deployable Cover senior project [4]. They designed an aluminum flexure with a very small bond area that broke under the small loads of just handling the mirror. As a preliminary test to ensure there weren’t any egregious mistakes in this design a small load was applied to the invar. Both pulling the invar radially outwards as well as tangent to the mirror. Both the top and bottom mirrors passed this test that they can at least sustain the loads of assembling the mirror into the vibration test fixture and later into the instrument. With this they are ready to be used in testing the flexures on the shake table.

4.4 - Vibration Testing
The purpose of vibration testing is to experimentally validate the natural frequencies modeled using finite element analysis by correlating the analytical model with real world test data. The second purpose of vibration testing is to qualify the system for spaceflight per NASA’s GEVS standard for flight hardware [14]. Due to manufacturing delays, only the top mirror flexures were manufactured and tested. The vibration testing consists of two main tests, sine sweep and random vibration input. The purpose of the sine sweep is to experimentally determine the natural frequencies of the system and is also used to determine if anything shifted before and after the random vibration test. The purpose of the random vibration input is “to verify its ability to survive the lift-off environment and also to provide a final workmanship vibration test” [14]. Tests consists of running the three top mirror flexure geometries on a sine sweep to determine the natural frequencies between 5 and 2000Hz, followed by the random vibration input testing and concluded with a sine sweep test to see if any modes shifted during the random vibration testing. The random vibration testing is performed first at low input and slowly increasing the input level until reaching the full input curved designed in NASA’s GEVS standard for random vibration testing. Each flexure shall be tested along all three mutually orthogonal directions, X-axis, Y-axis and Z-axis. Refer to Appendix 5 for the full testing procedure.
4.4.1 - Vibration Testing Methods  
The methodology of the vibration testing performed was to follow the industry standard as set by NASA’s General Environmental Verification Standards and of that, the sections that pertain to sine sweep and random noise. All vibration testing was conducted at Cal Poly’s Mechanical Vibrations lab where there are two controllable shake tables. The shake tables are Ling Electronics A395 and L612 models controlled by a Crystal Instruments controller. The L612 shaker was in the horizontal configuration with an attached slip table for horizontal mounting and the A395 table was in the vertical configuration. The Shake Tables can take in up to four input accelerometers and the controller will output the acceleration data as raw data as well as a frequency response plot. Both shake tables have a frequency range of 5Hz to 3000 Hz although the upper limit for this project’s testing was 2000Hz.

Both tables perform similarly and have a very similar interface from software with the only meaningful difference being the mechanical mounting. The vertical shake table has a radial bolt pattern where the bolt holes are in a 3” ring spacing and a 6” ring. Conversely the horizontal slip table has a 3” on center square bolt pattern. The only impact this had was the need for a universal adapter plate that could mount to each shake table.

For all tests, two single axis VIP Sensors Model 1011 piezo-electric accelerometers were used. One mounted to the base and one mounted to the mirror to measure the response of the mirror compared to the base. The Crystal Instruments controller tags in a calibration constant for each accelerometer used in mV/g. As these accelerometers are piezo-electric, they output a signal of pC/g which had to be converted to mV. These accelerometers are then connected to a charger which has a constant conversion of the signal of 10mV/pC. This was then connected to a power supply that amplifies the signal and sends it to the vibration controller.
Although this setup of the accelerometers was consistent throughout testing, the method of mounting to the mirrors evolved over time to mount to the mirrors and base more rigidly. The first method used in preliminary testing was Petro Wax, a red waxy material that is very sticky and does well for temporary mounting but is not rigid enough for the extreme vibrations of the random vibration tests. The next method was to super glue the accelerometer to the backside of Kapton Tape and stick the tape onto the mirror. This worked well but after multiple applications the tape would lose its adhesive and become a weaker bond. This method was used in preliminary testing, but for the final rounds of all testing, double-sided Kapton Tape was used to mount the accelerometer to the mirror and the base. This method had the same benefits of the superglue method without getting glue onto the accelerometer itself. See below figure of the three methods. The other mounting change was the placement of the accelerometers. The original placement for the preliminary testing was to place the accelerometer on top of the mirror in the direction of the test. As these accelerometers are single axis, the alignment is critical for good measurements. Mounting on top of the mirror posed two challenges. The first challenge was that it was difficult to ensure it was perpendicular to the mirror plane and in line with the direction of travel. The second challenge was that although the accelerometer is low mass, having the accelerometer off the line of the CG could allow for slight bending with relation to the axis of motion. To mitigate both of these flaws, the accelerometer was mounted to the side of the mirror so that it was in line with the CG and more reliably in line with the motion so that the single axis accelerometer could read the full signal during testing and there would be no off-axis effects.
Figure 63. Petro Wax mounting method (top left), Super Glue and Kapton Tape method (top right), and double sided Kapton Tape (bottom center)

4.4.2 - Fixture Vibration Testing
Preceding testing of any of the flight hardware, a baseline test of the mounting fixture was performed. This testing follows the same procedures as what is performed for testing the flexures. The empty fixture was tested on the same shake tables and served to verify the shake table and controller software worked as intended before any flight hardware was tested.
Figure 64. Empty Mounting Fixture on shake table for preliminary testing

Figure 65. Vibration Mounting Fixture sine sweep in the horizontal axis
Figure 66. Vibration Mounting Fixture sine sweep in the vertical axis

As shown in the above plots, the natural frequency of the test fixture lies outside that of the frequency range used for testing. The sine sweep test and random vibration test were conducted for the fixture and the results show that there is no concern that the fixture will have a natural frequency near that of the test articles. The fixture showed nearly no response across the whole 5Hz to 2000 Hz frequency range as evidenced by the lack of any deviation from the blue shake table base response. This testing was done to ensure and verify there would not be mode coupling of the flexures and mounting plate wherein the two have close natural frequencies and when excited, they amplify each other. The natural frequencies of the flexure systems are in the 100-500 Hz range compared to that of the mounting plate which is in the kHz range, this difference is sufficient to not cause mode coupling and means the flexure will be a good test bed for flexure testing.
4.4.3 - Autocollimator Testing

While the vibration testing will qualify the overall design, there is concern that the mirror could shift through testing and lead to inoperable optical elements. The optical assembly as a whole has a 25 arc-minute tolerance so testing must be performed to verify that the mirrors shift less than that through all testing. The Autocollimator is a device used to measure the angular shift of an optical assembly. It works by shining a light down a tube at the mirror which reflects at the eye piece. Inside the autocollimator there is a crosshair used to align the tool and then after the vibration test, used to measure the amount of shift in the system. The autocollimator is adjusted to a “zero” position with two axes of angular adjustment to set the home location before the vibration testing. After testing, the mirror is placed under the autocollimator and the misalignment of the cross hairs is a measurement of the angular displacement in two axes. It measures the roll and the pitch of the mirror, compared to its starting position before testing.

![Autocollimator Diagram](image)

**Figure 67.** Autocollimator Diagram [27]

The autocollimator is mounted to a breadboard and a 45° angle mirror is used to direct the light down to the mirror. The mirrors and flexures are in turn mounted to the vibration table mounting plate. The below figure shows the autocollimator setup. For repeatability of measurements, a 3-2-1 mounting scheme was implemented to ensure that each mirror was reliably measured. The flat plate of the breadboard formed the plane that the mounting plate would rest on. Two mounting points on one axis were utilized and on the perpendicular axis a single ball point mount was used. This 3-2-1 mounting scheme ensures the
mounting fixture is placed on the breadboard in the exact same location every time. This is important so that when measuring the change in angle of the mirror, only the shift of the mirror is what should be measured, not any changes in alignment of the fixture.

![Autocollimator Mounting Setup](image)

**Figure 68.** Autocollimator Mounting Setup

Prior to performing any vibration testing on the shake table, the baseline of the mirror is established. The mirror is placed on the breadboard as shown in the figure above and the alignment knobs of the 45° angle mirror are adjusted to get the flexure/mirror system to align with the cross hairs of the autocollimator. The below figure shows what it looks like to look into the eyepiece once the autocollimator is calibrated. To ensure that the measurement is reliable, the mounting plate with mirror is removed and replaced several times to test the repeatability of placing the mirror under the autocollimator. Through this, the total variation of placing the mounting plate under the autocollimator is only 1-2 arc minutes, an acceptable amount for given the allowable of the total mirror system is an order of magnitude higher.
4.4.4 - Curved Blade Vibration Testing

The curved blade was analyzed to have three modes within the 5-2000Hz range that the sine sweep test passes through. During analytical modeling of the curved blade, the first three modes were found to be on the three mutually orthogonal axes. Experimental testing in all three principal axes experimentally verifies the three modes seen in analysis.

The vertical, Z axis, was analyzed to have its first natural frequency at 395 Hz with no other natural frequencies in that axis within the 5Hz – 2000Hz frequency band that the hardware is tested in. The curved blade was mounted to the vertical shake table using a custom designed mounting plate and an adapter plate. The mounting plate serves to replicate the bolt pattern the flexures mate with on the MUVI instruments doors and the adapter plate is designed to mount to both the vertical shake table and horizontal shake table’s mounting bolt patterns.
The first test to be completed in the vertical configuration was the sine sweep test as described in NASA GEVS standard section 2.4.3. Followed by the vibroacoustic qualification performed via the Random Vibration Input test as described in GEVS section 2.4.2. The below images show the test setup on the vertical shake table as well as the experimental results from each test.

**Figure 70.** Vertical Shake Table test setup

![Vertical Shake Table test setup](image)

**Figure 71.** Curved Blade Flexure Vertical Axis Sine Sweep - pre and post overlay

![Curved Blade Flexure Vertical Axis Sine Sweep - pre and post overlay](image)
The results of the pre and posttest sine sweep show that the experimental natural frequency of the Curved Flexure Blade is 381Hz. This is a 3.5% difference between the experimental and analytical values. Additionally, the random vibration testing fully qualifies the design to the Qualification curved as defined in section 2.4.2 of NASA GEVS. The curved blade was ran up to the full 0.16 g²/Hz Amplitude Spectral Density (ASD) level as required and no failures of the flexure were noted. This is further backed up by the limited to no shift of the natural frequency in the post test sine sweep as well as the marginal shift in reading on the autocollimator.

Following the testing on the vertical shaker table, the mirror-flexure system was mounted to the horizontal shaker for testing in the X and Y axes. Starting with the x-axis, the flexure went through the pre-test sine sweep, random vibration input and post-test sine sweep followed last by the autocollimator visual verification. The plots below show the overlay of pre and posttest sine sweeps showing little to no change in the overall system throughout the random vibration qualification testing. A shift in natural

![Graph](image-url)

**Figure 72.** Curved Blade Flexure Vertical Axis Random Vibration Input
frequency or overall shape would indicate something structural had changed during the random vibration testing. With nearly identical natural frequencies and overall shape, the sine sweep test shows that the curved blade was well qualified by the random vibration testing.

**Figure 73.** Horizontal Shake Table test setup

**Figure 74.** Pre and Post Sine Sweep Horizontal X Axis - Curved Blade Flexure
During the Random Vibration Test in the horizontal axis, there was a limitation of the shake table power amplifier that it could only take the specimen up to 60% of the full power curve. Although this does not meet the full power test objective, this still collects good data indicating the flexure is robust and well qualified to flight loads. The NASA GEVS standard that the test case was following is the 2x qualification curve as this is prototype hardware. For the flight hardware, the acceptance test case is half of the qualification curve and the testing here shows that the curved blade is well bounded for acceptance testing.

**Figure 75.** Random Vibration Horizontal X Axis - Curved Blade Flexure

Following the X-axis testing, the assembly was rotated 90 degrees to re-test all test cases in the final principal axis. Testing followed the same procedure and plots of the test data are below. As indicated in the sine sweep overlay plot, the curved blade was again validated to meet the qualification test as evidenced by the near identical pre and post-test natural frequencies.
**Figure 76.** Pre and Post Sine Sweep Horizontal Y Axis - Curved Blade Flexure

**Figure 77.** Random Vibration Horizontal Y Axis - Curved Blade Flexure
Through testing the curved blade in all axes according to the NASA GEVS standard qualification test for random vibration input, the curved blade was proven to pass all test cases and is qualified for flight. The pre and posttest autocollimator results also show little change in angular position of the mirror indicating the design is suitable for this application as the mirror shift of approximately 5 arc minutes at the high end is within the acceptable range. Through the extensive testing of the curved blade up to the full qualification input levels, the design is proven to be robust and will work well for the application in the MUVI instrument. Additionally, the minor shift between analytical modeling and experimental results indicates the finite element model is both accurate and robust.

### 4.4.5 - Bent Blade Vibration Testing

Testing of the Bent Flexure blade design followed the same procedure, sine sweep and random vibration input in all three axes. To start, the vertical axis was tested running the flexure on a low input sine sweep from 5 Hz to 2000Hz. The analytical model showed the natural frequency would be at 212 Hz. Through testing in the vertical axis, the experimental natural frequency of 196 Hz was determined. Following the initial sine sweep, random vibration test cases were performed, ramping up from -12dB first pass up to the full level per the test standard. Through this testing a similar natural frequency was observed, and the overall design was qualified for this axis. A post-test sine sweep was performed with minimal change in peak response and natural frequency. Finally, the posttest autocollimator verification was within the expected bounds of 5 arc minutes. The below plots show the overlay of the pre-test and post-test sine sweeps and the random vibration test cases from -12dB up to 0dB (full power).
**Figure 78.** Pre and Post Sine Sweep Vertical Axis - Bent Blade Flexure

**Figure 79.** Random Vibration Vertical Axis - Bent Blade Flexure
With the vertical testing complete, the horizontal testing was next up on the test stand. Testing began with a sine sweeps in both the x and y axis at low input levels to compare the analytical value of 203 Hz for each axis to the real world. The low-level sine sweep test determined an experimental natural frequency in the x-axis of 247 Hz and 253Hz for the y-axis.

Following that the random vibration testing was performed, starting with the x-axis. Test cases were started at the low -12dB power level and walking up towards the full power level. When at the -3dB power level, the flexure failed, and the test ended. The failure happened right in the 90-degree bend of the flexure blade. As this blade had previously failed before the shoulder bolts were added, this failure could not be rationalized as being due to the flexures slipping in relation to their fasteners as had been presumed when shoulder bolts were not used. This failure indicates that the bent blade has a stress concentration in these corners and cannot be qualified to the vibrational energy required to meet the GEVS specification. As failures were observed in this axis with both standard fasteners and shoulder bolts, the root cause was determined to be the flexure itself and not shock to the system from slipping due to a lack of pre-load. The bent blade testing was concluded, and the final determination was that this design was not qualified.
Figure 80. Pre-Test Sine Sweep Horizontal X Axis - Bent Blade Flexure

Figure 81. Random Vibration Horizontal X Axis - Bent Blade Flexure
Table 22. Bent Blade Natural Frequency Comparison

<table>
<thead>
<tr>
<th>Mode</th>
<th>Axis</th>
<th>Analytical Frequency [Hz]</th>
<th>Experimental Frequency [Hz]</th>
<th>Percent Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Z</td>
<td>196</td>
<td>212</td>
<td>8.2%</td>
</tr>
<tr>
<td>2</td>
<td>X</td>
<td>203</td>
<td>247</td>
<td>21.8%</td>
</tr>
<tr>
<td>3</td>
<td>Y</td>
<td>203</td>
<td>253</td>
<td>24.6%</td>
</tr>
</tbody>
</table>

Although the testing result was not what was desired, a failure of the design is good data and shows the importance of real-world vibration testing. The bent blade, with its sharp 90-degree bends always had the concern that it could be a stress concentration. In the design phase, the tradeoff of having the smaller packaging that the bends afforded as well as the extra compliance were accepted even though the risk of the corners being a stress concentration was expected. The failure of the bent blade shows the importance of testing multiple designs in the prototype phase as not all designs behave in the real world as they do in analysis. A notable finding was that in the horizontal configuration, the designs had significantly high natural frequencies, 22-25% higher than analysis predicted. This indicates that the structure was much more rigid than had been anticipated and in turn more load was going into the flexures.
4.4.6 - Z Blade Vibration Testing
The final sets of testing to be performed were the Z Blade flexures. Starting, with the horizontal shake table the blades underwent the sine sweep and random noise tests. Results showed that the Z Blade had an experimental natural frequency of 163 Hz compared to the analytical model with a natural frequency of 150 Hz. The flexure assembly went through the random vibration tests, ramping up from the -12dB power level up to the full power level with positive results from the test. Following the random vibration tests, the post-sine sweep test was performed and the natural frequency was again observed to be 163Hz. This lack of shift in natural frequency following the random vibration test validates that nothing shifted or otherwise failed during the test and therefore the flexure passed in this orientation. This is visually seen in the pre-test and post-test sine sweep overlay plot below. Additionally, a minor 4 arcmin shift in the mirror was recorded on the autocollimator, which is within the resolution of the test, and indicates a passing result for qualifying the flexure in the vertical axis.

![Pre and Post Sine Sweep Vertical Axis - Z Blade Flexure](image)

**Figure 83.** Pre and Post Sine Sweep Vertical Axis - Z Blade Flexure
Following the vertical axis testing, the Z Blade was mounted to the horizontal shake table, oriented in the X axis and the same suite of testing was performed. In this axis, the analytical natural frequency was 123Hz and the pre-test sine sweep recorded the first mode at 147 Hz. The random test was then completed, following the ramp up procedure from -12 dB to full power with positive results for the flexure. The post-test sine sweep did show a minor shift in natural frequency as seen in the plot below. The pre-test natural frequency was 147Hz and the post-test natural frequency was recorded to be 143 Hz, or a 2.7% decrease. Although the natural frequency was slightly lower, the post-test autocollimator measurement did not indicate a significant physical shift of the mirror assembly.

**Figure 84.** Random Vibration Vertical Axis - Z Blade Flexure
**Figure 85.** Pre and Post sine sweep Horizontal X Axis - Z Blade Flexure

**Figure 86.** Random Vibration Horizontal X Axis - Z Blade Flexure
The last test up was the Y axis for the Z blade. The fixture was rotated 90 degrees to test in the orthogonal axis and testing began with the sine sweep. For the Y axis, the analytical natural frequency was 123 Hz as well and the pre-test sine sweep recorded a natural frequency of 151 Hz. Moving next into the random vibration testing, the flexure went through -12 dB, -9dB, and -6dB power levels with no abnormal test signatures or responses. When stepping into the -3dB power level, the flexure the is located directly in line with the axis of motion snapped right at the sharp bent in the blade.

![Y-axis Random Vibration](image.png)

**Figure 87. Z Blade Flexure Testing Failure**

![Pre-test Sine Sweep Horizontal Y Axis - Z Blade Flexure](image.png)

**Figure 88. Pre-test Sine Sweep Horizontal Y Axis - Z Blade Flexure**
Table 23. Top Mirror Z Blade comparison of analytical and experimental results

<table>
<thead>
<tr>
<th>Axis</th>
<th>Analytical Frequency [Hz]</th>
<th>Experimental Frequency [Hz]</th>
<th>Percent Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>123</td>
<td>151</td>
<td>22.8%</td>
</tr>
<tr>
<td>Y</td>
<td>123</td>
<td>147</td>
<td>19.5%</td>
</tr>
<tr>
<td>Z</td>
<td>150</td>
<td>163</td>
<td>8.7%</td>
</tr>
</tbody>
</table>

Much like the Bent Blade design, the failure of the flexure was directly at the sharp corner where there is a stress concentration. Additionally, the blade that failed is the one that is unsymmetrically loaded. That is one drawback of the tri-mounting design, is that in some orientations, one blade takes more load than the other two and that is what occurred here. Although the testing result was not what was desired, failure of the design is important information into the design and overall good for the MUVI project as an improperly designed part at the individual level will not make its way into the full integrated assembly. This failure shows the importance of real-world vibration testing and the importance of qualifying parts at the part level and not relying wholly on assembly level validation. Much like the Bent Blade, the sharp angles in this design were a watch item in testing due to stress concentrations. Still the design was pushed forward into testing due to its uniquely low natural frequency and therefore compliant design. The failures
of both the Bent Blade and Z Blade highlight the importance of testing multiple designs in the prototype phase as not all designs behave in the real world as they do in analysis.
Chapter 5 - Conclusion
Throughout the course of this project, to develop a mechanical mounting assembly for the MUVI optical system, the overall goal was to qualify a design for use. As outlined previously, the qualification was to meet that of the NASA GEVS specification for qualifying mechanical assemblies for applications in space. NASA GEVS’s exacting standards ensure that all designs will be thoroughly tested and qualified to fly on any launch vehicle. With that, throughout the extensive test campaign for this design, the curved blade came out as the front runner and is the best design for the mirror and MUVI as a whole. The curved blade passed all qualification testing per NASA GEVS in all orientations. This testing well bounds any environment the flexure will see during launch and when in service in space.

Several launch vehicles publish the acoustic and random vibration environment and these show that the GEVS testing procedure bounds the random vibration environments very well. For example, per SpaceX’s Falcon 9 Payload User Guide [1] the power spectral density curve of Falcon 9’s random vibration environment is much lower than the qualification curve from NASA GEVS. The figures below show F9’s random vibration environment compared to the NASA GEVS qualification standard.

![Figure 90. Falcon 9 Random Vibration Environment][28]
Figure 91. NASA GEVS Random Vibration Qualification curve

The area under each curve is the measure of how much total acceleration energy is going to the hardware and is a good way to compare the two curves. As both curves are covering the same frequency spectrum from 20 Hz to 2000Hz, the area under each curve ($G_{RMS}$) compares to total energy. For Falcon 9’s curve, hardware experiences an overall 5.13 $G_{RMS}$ compared to the 14.1 $G_{RMS}$ on the GEVS qualification curve and 10.0 $G_{RMS}$ on the acceptance curve. Based on the 63% reduction in $G_{RMS}$ loading when flying on Falcon 9, the Curved Flexure Blade is well qualified to fly.

Figure 92. Falcon 9 maximum random vibration predicted environment.
Furthermore, the significantly lower $G_{RMS}$ loading that payloads experience on Falcon 9 compared to the inputs prescribed by GEVS qualification testing procedure, all flexure blades would be suitable for flight. During the random vibration testing of each flexure, all blades could withstand the -3dB random vibration input which has a $G_{RMS}$ of 7.05. Although all flexures might be acceptable for flight, that does not mean they are qualified. Generally, when qualifying a prototype for flight, the qualification campaign goes above and beyond what will be experience in flight. As the curved blade was the only one to be fully qualified, the recommendation of this paper is to use the Curved Flexure Blade for MUVI.

With all testing complete, the below summary tables the comparison of the analytical results and experimental results. In most cases the analytical results and the experimental agreed, for example the curved mirror showed only a max of a 5.3% difference between the analyzed and the real world. However, the Bent Blade and Z Blade designs differed quite a bit more, up to a 24.6% difference. The more complex geometry, with the sharp corners and tight profile tolerances led to a more challenging shape to model. Additionally, these were the flexures that failed in testing, indicating that the analysis under predicted the loads they would see in testing. As shown in the tables below the Bent Blade and Z Blade analysis consistently predicted a lower natural frequency than was seen in testing. This indicates that the finite element analysis was modeling a more flexible and compliant system than was real. This could be a contributing factor to why these were the flexures that were seen to fail in testing.

![Figure 93. GEVS random vibration qualification and acceptance profiles.](image-url)

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Qualification</th>
<th>Acceptance</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>0.026</td>
<td>0.013</td>
</tr>
<tr>
<td>20-50</td>
<td>+6 dB/oct</td>
<td>+6 dB/oct</td>
</tr>
<tr>
<td>50-800</td>
<td>0.16</td>
<td>0.08</td>
</tr>
<tr>
<td>800-2000</td>
<td>-6 dB/oct</td>
<td>-6 dB/oct</td>
</tr>
<tr>
<td>2000</td>
<td>0.026</td>
<td>0.013</td>
</tr>
<tr>
<td>Overall</td>
<td>14.1 $G_{rms}$</td>
<td>10.0 $G_{rms}$</td>
</tr>
</tbody>
</table>
Table 24. Top Mirror Bent Blade comparison of analytical and experimental results.

<table>
<thead>
<tr>
<th>Axis</th>
<th>Analytical Frequency [Hz]</th>
<th>Experimental Frequency [Hz]</th>
<th>Percent Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>196</td>
<td>212</td>
<td>8.2%</td>
</tr>
<tr>
<td>X</td>
<td>203</td>
<td>247</td>
<td>21.8%</td>
</tr>
<tr>
<td>Y</td>
<td>203</td>
<td>253</td>
<td>24.6%</td>
</tr>
</tbody>
</table>

Table 25. Top Mirror Curved Blade comparison of analytical and experimental results.

<table>
<thead>
<tr>
<th>Axis</th>
<th>Analytical Frequency [Hz]</th>
<th>Experimental Frequency [Hz]</th>
<th>Percent Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>395</td>
<td>374</td>
<td>5.3%</td>
</tr>
<tr>
<td>X</td>
<td>544</td>
<td>531</td>
<td>2.4%</td>
</tr>
<tr>
<td>Y</td>
<td>544</td>
<td>536</td>
<td>1.5%</td>
</tr>
</tbody>
</table>

Table 26. Top Mirror Z Blade comparison of analytical and experimental results.

<table>
<thead>
<tr>
<th>Axis</th>
<th>Analytical Frequency [Hz]</th>
<th>Experimental Frequency [Hz]</th>
<th>Percent Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>123</td>
<td>151</td>
<td>22.8%</td>
</tr>
<tr>
<td>Y</td>
<td>123</td>
<td>147</td>
<td>19.5%</td>
</tr>
<tr>
<td>Z</td>
<td>150</td>
<td>163</td>
<td>8.7%</td>
</tr>
</tbody>
</table>

Table 27. Autocollimator Results

<table>
<thead>
<tr>
<th>Flexure</th>
<th>Axis</th>
<th>Angular Displacement [arc min]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Curved Blade</td>
<td>X</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>2.5</td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>3</td>
</tr>
<tr>
<td>Bent Blade</td>
<td>X</td>
<td>N/A</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>N/A</td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>1</td>
</tr>
<tr>
<td>Z Blade</td>
<td>X</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>N/A</td>
</tr>
<tr>
<td></td>
<td>Z</td>
<td>4</td>
</tr>
</tbody>
</table>

The table above shows the results of the autocollimator displacement measurements. As stated in the beginning of the project, the goal was to not only design a system to mechanically isolate the mirror from the vibration of launch but to also maintain tight angular tolerances for the alignment of the optics in the
instrument. The goal at the outset was to see approximately 1 arc min of angular displacement or less but due to the tolerances allowed by the autocollimator, the requirement was relaxed to 5 arcmin for the purposes of qualification testing. The method was to zero or “tare” the measurement device to the mirror prior to each test, perform the sine sweep and random vibration test for a given axis and re-measure the angular displacement after testing. As shown in the table above, the Curved Blade met this requirement on all axes. The other two flexure designs were passing for the axes that had passing vibration testing but those that failed in testing were not measured. The figure below shows the Curved Blade autocollimator results.

![Autocollimator results - Curved Blade](image)

**Figure 94.** Autocollimator results - Curved Blade: X-axis, Y-axis, Z-axis (right to left)

Regarding the two flexures that failed in testing, the Bent Blade and the Z Blade, there were several root causes that were determined to play a role. At first, the assumption was that the flexures were slipping in the mount to the adapter plate due to insufficient preload. The original fasteners used were standard M2.5 fasteners, designed for tension only use cases. After the preliminary testing showed these were insufficient, shoulder bolts were designed so that the fasteners could take shear loads. More detail on the shoulder bolt is in the following chapter. The second round of testing saw failures of some fasteners...
losing preload and backing off. This was corrected by using an epoxy fillet to act as secondary retention on the fastener and prevent them from backing off. After both of these were corrected, the two designs still failed and left with the question of why.

Revisiting the two designs after testing revealed that they were undersized in the original designs. Buckling analysis was performed on the individual blades themselves and showed them to be susceptible to buckling in the direction tangent to the mirror. Buckling analysis imposes a 1N “unit load” to the flexure at the point where the mirror mounts. It is loaded in the three primary directions that were tested and a buckling factor of 8.1 was analyzed for the Z Blade. This means that the Z Blade would buckle at 8.1N of load to it in the tangential direction. Going back to the random vibration testing performed on the Z Blade, a peak acceleration of 399G was measured at just the 50% random vibration input level when in the Y-Axis configuration. This with the 62g mass of the mirror equates to 24.7N. The individual flexure was analyzed to buckle at 8.1N and the 24.7N load is shared across the three blades, and two of the blades are out of the direction that is sensitive to buckling, but it is reasonable to conclude that the poor buckling characteristics of the flexure lead it to fail prematurely. The below figure shows the orientation of the flexure when it failed, clearly in the configuration where it was loaded purely in the tangential direction that is sensitive to buckling. The same analysis was performed for the bent blade and showed the same root cause as being sensitive to buckling in addition to the inherent risk of designing with sharp corners that are known to have stress concentrations. With the root cause determined, the following chapter on future work for MUVI discusses corrective actions to update the design of the flexures such that they do not fail in testing.

<table>
<thead>
<tr>
<th>Load [N]</th>
<th>Direction</th>
<th>Buckling Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>Normal</td>
<td>16.2</td>
</tr>
<tr>
<td>1.0</td>
<td>Tangential</td>
<td>8.1</td>
</tr>
<tr>
<td>1.0</td>
<td>Radial</td>
<td>16.2</td>
</tr>
</tbody>
</table>
In summary, the Curved Blade design is the overall winner as it met all the design requirements that were established at the start of the project. The Curved Blade flexure met the requirements of mass and size, natural frequency as well as passing the qualification testing required to vet the hardware for spaceflight. Although the thermal aspects of the project were analyzed, there was no explicit thermal testing to validate the design of using Invar as intermediate material to bridge the gap in CTE. Future system level testing of the thermal aspect of MUVI will experimentally validate the thermal analysis but as that was not a driving load case it is not of concern that it would invalidate the results of having the Curved Blade as the final selected design for the flexures.

5.1 - Lessons Learned
Throughout the design, analysis, and predominantly the manufacturing and testing phases of the project, several important lessons were learned. As it pertains to design and analysis the biggest lesson learned was to keep the design simple. This also plays into manufacturability, the complexity of these parts meant for long lead times and limited supplier availability. The bulk of the lessons learned were when
everything came together, was integrated and tested. These include paying close attention to tolerance stack-ups, bolted-joint interactions and the type of fasteners.

The flexures were designed with the constraints of a small design envelope, low mass, and the requirements of isolating the mirrors. This led to the design of very small and thin blades with complex geometry. Additionally, titanium was selected due to its high strength to weight ratio. This did have the downsides of considerably increasing the complexity of manufacturing. The very small size and tight tolerances of the parts meant that wire electric discharge machining (EDM) was required for manufacturing. This is an expensive, time-consuming process that has only a few vendors nationwide that can accomplish it. As a result, this project saw considerable delays working with outsourced vendors and also led to very high machining costs for these prototypes. In retrospect, a more traditional manufacturing process such of CNC machining would have been a much better process. Although the design met other requirements, the design for manufacturability (DFM) should have had a higher weighting in the original design requirements.

The other lesson learned regarding manufacturing was the criticality of dimensional tolerancing between assemblies. During design, two tabs were added to the invar bonding pieces to mitigate the possibility of the mirror rotating. These anti-rotation tabs were modeled to be the same width and the square feature on the flexures. The problem being, this required both the flexure and the invar to be machined to exact tolerances, even though the drawing allowed ±0.005”. The tolerance stack-up in this location was overlooked during the design phase and lead to issues during integration. To solve this stack-up issue, the titanium of the flexures was hand sanded down with very fine 600 grit sandpaper until proper fitment was achieved. In total the parts only interfered by a couple thousandths of an inch, but this was enough to cause fitment issues. Sanding down a couple thousandths of an inch was a suitable repair but should have been caught before parts were sent out for manufacturing to make the integration process go smoother.
During initial testing, multiple failures of the flexures were observed and ultimately corrected but could have been anticipated during original design. The most catastrophic failure during initial testing was breaking flexure blades themselves. The root cause for these failures was determined to be the fasteners used to mount to the base plate. Standard M2.5 fasteners were selected and the through holes on the flexures were sized to the standard through hole size for said fasteners. The problem that this created was that in the horizontal vibration test configuration, there was nothing resisting shear and this allowed the fasteners to shift violently during random vibration testing. The sudden shift due to lack of fasteners resisting side to side motion acted as a shock input to the system and was enough to fracture the flexures.

To mitigate this shear problem in the horizontal direction, custom shoulder bolts were designed and fabricated. As the flexures were already manufactured, off the shelf shoulder bolts could not be used as the size of the flexures’ through hole didn’t align with standard shoulder bolt sizes. To solve this problem during testing, pin gauges were used to empirically measure the hole diameter of all flexures down to 0.001” tolerance and then shoulder bolts were custom fabricated. This did work successfully to mitigate the shear movement issues seen in testing but in hindsight, should have been designed from the start.
Designing the flexures with through holes that match standard sized shoulder bolts would have been a much more robust design and make part procurement much easier.

Figure 97. Comparison of fastener clearance in the flexure through hole. Standard M2.5 fastener (left) custom shoulder bolt (right)

The final lesson learned through testing was that the high vibration environment that the assembly experiences during random vibration can cause fasteners to back off if not fully torqued down. Even with proper fastener torque application, this failure of fasteners backing out is commonly observed in the high vibration environments of aerospace. The industry standard is to utilize secondary retention such as safety cable, locking patch fasteners or staking fasteners. During random vibration testing, a fastener was able to
fully back itself off and this invalidated the test. To mitigate this for all future tests, epoxy was used to stack the head of the fasteners. This process of staking the heads applies a bead of epoxy to the fastener head and essentially bonds the head in place so that it cannot back out. In addition to staking all fasteners, torque stripes were added to visually verify if fasteners were backing out. A torque stripe is just a line on the fastener head and base metal, usually done with a Sharpie or similar, to visually show that the joint is intact. If the fastener backs off, the stripe will be broken, and it will be easy to visually inspect. During all random vibration testing, torque stripes were applied to all joints and inspected between rounds as the input was increased to look for any fasteners that might have backed off. Through all testing, the staking of fasteners worked flawlessly, and no fasteners backed out.

**Figure 98.** Illustrative image of epoxy staking fasteners
5.2 - Future for MUVI
At the conclusion of this project, a front runner design for the flexures that mount the mirror has been determined. Through the testing of the hardware and the failures observed, there are some design updates to be made before the flexures are fully flightworthy. As mentioned in the conclusion chapter, the Curved Blade is qualified for flight based on NASA GEVS load profiles, but the Bent Blade and Z Blade are not yet qualified. If those designs were to be pursued, some minor updates would need to be made. Namely, some small increases to the overall thickness of the blade and height to allow for stronger design while maintaining the majority of the compliance. The thicker geometry will aid in increasing the ability of the flexure to take loads in directions that are sensitive to buckling and overall make for a stronger design. Additionally, if redesigning the flexures and a new batch were to be manufactured, it is strongly recommended to account for commercial off the shelf shoulder bolts. This was a change based on the finding in testing and custom machined shoulder bolts were procured to continue with testing but if starting on a revision 2 of the flexures, taking into account shoulder bolts would be highly recommended. For example, McMaster Carr sells PN: 90265A323, a M2.5 shoulder bolt with a shoulder length of 12mm and diameter of 3mm. This would take very minor changes to the flexure mounting design and result in the ability to utilize commercial off-the-shelf parts.

In addition to some remaining redesign work needed for the flexures based on the outcome of the testing and the lessons learned, there is one other remaining project for MUVI. This project is a design of the exterior coating of the instrument for thermal control and is still in work at this time. Once that is complete, MUVI will be ready for system level integration and testing. This will be the first test of putting together all the subsystems and running through all the testing required to qualify the instrument. MUVI as a system will undergo the same vibration testing as the flexures went through and then the rigorous thermal vacuum (TVAC) testing to put the whole system through the space like environments in a near vacuum through a wide range of temperatures. With the completion of system level testing, the MUVI instrument will be ready for satellite bus integration and launch.
BIBLIOGRAPHY


[24] Eight-node brick element (C3D8 and F3D8), [Online], Available, https://web.mit.edu/calculix_v2.7/CalculiX/ccx_2.7/doc/ccx/node26.html#:~:text=The%20C3D8%20element%20is%20a,numbered%20according%20to%20Figure%2054 (Accessed: March 2, 2021).


APPENDICES

A. Mechanical Drawing Package

<table>
<thead>
<tr>
<th>Part No.</th>
<th>Title</th>
<th>Qty</th>
</tr>
</thead>
<tbody>
<tr>
<td>MXY-FLEX 01</td>
<td>Top Bent Pressure Blade</td>
<td>6</td>
</tr>
<tr>
<td>MXY-FLEX 02</td>
<td>Top Curved Pressure Blade</td>
<td>6</td>
</tr>
<tr>
<td>MXY-FLEX 03</td>
<td>Top &amp; Pressure Blade</td>
<td>6</td>
</tr>
<tr>
<td>MXY-FLEX 04</td>
<td>Top Linear Pad</td>
<td>12</td>
</tr>
<tr>
<td>MXY-FLEX 05</td>
<td>Top Door Swing Blade</td>
<td>2</td>
</tr>
<tr>
<td>MXY-FLEX 06</td>
<td>Top Door - Curved Blade</td>
<td>2</td>
</tr>
<tr>
<td>MXY-FLEX 07</td>
<td>Top Door - Z Blade</td>
<td>2</td>
</tr>
</tbody>
</table>

NOTES - UNLESS OTHERWISE SPECIFIED:
1. ALL DIMENSIONS ARE IN MILLINDERS
2. INSTALL HORIZONTAL CODE INSERTS PER NAMSA 35537 REMOVE TANGS
3. TOLERANCES
   a. 0.005
   b. 0.005
   c. 0.005
4. ANGLES = +/-
1. ALL DIMENSIONS ARE IN INCHES
2. MATERIAL INVAR 36
3. QUANTITY 12
4. TOLERANCES
   X.XXX = .05
   X.XXX = .005
5. HOLE SIZE MILLING IN METRIC
6. INSTALL HELICAL COIL INSERTS PER NASM 33537.
   REMOVE TANG

<table>
<thead>
<tr>
<th>Part No.</th>
<th>Title</th>
<th>Qty.</th>
</tr>
</thead>
<tbody>
<tr>
<td>M0340001</td>
<td>Bottom Guide Fixture Blade</td>
<td>4</td>
</tr>
<tr>
<td>M0340002</td>
<td>Bottom Fixture Blade</td>
<td>4</td>
</tr>
<tr>
<td>M0340003</td>
<td>Bottom Fixture Blade</td>
<td>4</td>
</tr>
<tr>
<td>M0340004</td>
<td>Bottom Inner Blade</td>
<td>4</td>
</tr>
</tbody>
</table>

NOTES - UNLESS OTHERWISE SPECIFIED:
1. ALL DIMENSIONS ARE IN CHES
2. INSTALL HELICAL COIL INSERTS PER NASM 33537.
   REMOVE TANG.
3. TOLERANCES
   X.XXX = .05
   X.XXX = .005
   ANGLES = ±1
NOTES:
1. DETENT RADIUS .050 AND DEPTH .005

<table>
<thead>
<tr>
<th>Part No.</th>
<th>Name</th>
<th>Qty.</th>
</tr>
</thead>
<tbody>
<tr>
<td>MUV 401</td>
<td>Bottom Mirror</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>Assembly Base</td>
<td></td>
</tr>
<tr>
<td>MUV 402</td>
<td>Bottom Swirl Ring</td>
<td>1</td>
</tr>
<tr>
<td>MUV 403</td>
<td>Bottom Invar</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>Holder</td>
<td></td>
</tr>
<tr>
<td>MUV 404</td>
<td>Bottom Spring</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>Plunger Holder</td>
<td></td>
</tr>
<tr>
<td>MUV 405</td>
<td>Bottom Centering</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>Block</td>
<td></td>
</tr>
</tbody>
</table>

NOTES - UNLESS OTHERWISE SPECIFIED:
1. ALL DIMENSIONS ARE INCHES
2. MATERIAL OF CONSTRUCTION IS 6061-T6 ALUMINUM
3. TOLERANCES
   X XXX ± .005
   XXX ± .005
   ANGLES ± .1°
## B. Bill of Materials

<table>
<thead>
<tr>
<th>Group</th>
<th>Part Number</th>
<th>Description</th>
<th>Vendor</th>
<th>Unit Cost</th>
<th>Qty</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Top Door</strong></td>
<td>91292A836</td>
<td>M2.5X0.45 Socket Head, 15mm long</td>
<td>McMaster</td>
<td>5.34</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>93914A043</td>
<td>M2.5 Nitronic 60 Helical Insert 5mm long</td>
<td>McMaster</td>
<td>8.69</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>93914A009</td>
<td>M2.5 Nitronic 60 Helical Insert 2.5mm long</td>
<td>McMaster</td>
<td>7.53</td>
<td>7</td>
</tr>
<tr>
<td><strong>Bottom Door</strong></td>
<td>91292A012</td>
<td>M2.5X0.45 8mm long</td>
<td>McMaster</td>
<td>5.42</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>HCHM2545-004S40V</td>
<td>M2.5X0.45 4.5mm max vented</td>
<td>Lesker</td>
<td></td>
<td>25</td>
</tr>
<tr>
<td><strong>Top Assembly</strong></td>
<td>92855A843</td>
<td>M2.5x0.45 8mm long low profile</td>
<td>McMaster</td>
<td>12.83</td>
<td>3</td>
</tr>
<tr>
<td>Fixture</td>
<td>91292A012</td>
<td>M2.5x0.45 8mm long reg head</td>
<td>McMaster</td>
<td>5.42</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>8476A19</td>
<td>Spring Plunger</td>
<td>McMaster</td>
<td>5.62</td>
<td>6</td>
</tr>
<tr>
<td><strong>Bottom</strong></td>
<td>8477K28-8477K71</td>
<td>1.25&quot; dia glass plate</td>
<td>McMaster</td>
<td>9.5</td>
<td>1</td>
</tr>
<tr>
<td>Assembly</td>
<td>8477K28</td>
<td>3/8-16 bolts 3/4&quot; long</td>
<td>McMaster</td>
<td>11.69</td>
<td>1</td>
</tr>
<tr>
<td>Fixture</td>
<td>75045A78</td>
<td>3M 2216 Epoxy - Clear</td>
<td>McMaster</td>
<td>56.03</td>
<td>2</td>
</tr>
<tr>
<td><strong>Misc.</strong></td>
<td>EA9394 50ML</td>
<td>Loctite EA 9394</td>
<td>Ellsworth</td>
<td>35.38</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>75165A243</td>
<td>17GA Luer lock syringe tip</td>
<td>McMaster</td>
<td>11.88</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>75165A676</td>
<td>18GA Luer lock syringe tip</td>
<td>McMaster</td>
<td>12.43</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>46365A33</td>
<td>Scotch-Brite pads</td>
<td>McMaster</td>
<td>27.26</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>8567K52</td>
<td>0.005&quot; Mylar sheet 40&quot; x 10'</td>
<td>McMaster</td>
<td>22.34</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>7510A652</td>
<td>Luer lock syringe</td>
<td>McMaster</td>
<td>7.25</td>
<td>1</td>
</tr>
</tbody>
</table>
C. Vibration Test Procedure

MUVI Mirror Flexure Vibration Testing Procedures

Objective:
This document specifies step-by-step instructions to perform vibration testing on the MUVI Mirror Flexures. The purpose of physical vibration testing is to experimentally verify that the MUVI design adheres to NASA TRL 6 standards.

Tools and Equipment:
- Accelerometers (1x control mounted to Vibes Fixture Base Plate, 1x variable mounted on the center of the mirror, in line with the CG)
- Accelerometer adhesive: Kapton tape
- Vibes Test Fixture
- Mirror, Flexures and associated fasteners
- Shake Table mounting bolts
- PPE: Eye & ear protection

Reference Documentation
NASA-GSFC-STD-7000, General Environmental Verification Standard, Section 2.4

Test Summary
Vibration testing for MUVI involved placing accelerometers at various locations on the fixture and mirror and subjecting components to vibrations via a shake table. A control accelerometer will be placed on the vibes fixture where the flexures mount and a variable accelerometer will be placed on the mirror for every test. A third accelerometer will be used when the vibes fixture is in the vertical orientation for z-axis testing. In each configuration, the following tests will be performed in all three principal axis directions:
- Sine Signature Test
- Random Vibration Test (at -18, -12, -6, and 0 dB)
The sine signature test will be repeated after the random vibration tests to ensure fundamental frequencies did not change during random vibration.

Test Procedures for X axis
1. Bolt vibes fixture to shake table oriented in the X-axis
2. Bolt flexures and mirror assembly to vibes fixture
3. Record location and sensitivity of accelerometers used in experiment:

<table>
<thead>
<tr>
<th>Component</th>
<th>Accelerometer Location</th>
<th>Model</th>
<th>Serial Number</th>
<th>Sensitivity (mV/g)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mirror</td>
<td>Top Center</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vibes fixture</td>
<td>Top, away from mirror</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vibes fixture vertical</td>
<td>Top Center</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>adapter</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

4. Mount accelerometers
5. Tape accelerometer wires to prevent movement.
6. Sine Signature test
   a. Set or verify the following:
      i.  \( G \) Input level
      ii. 5-2000Hz
b. Perform sine sweep
c. Generate response plots (frequency vs amplitude) to assess fundamental frequencies.

7. Random Vibration test
   a. Set or verify the following:
      i. Accelerometer signals reading.
      ii. Accelerometer sensitivity correctly set.
      iii. Random profile as specified in GEVS correctly input.
      iv. Test duration correctly input.
      v. Input abort and alarm limits.
   b. Perform random vibration test at each level for at least 15 seconds each.
      i. -18 dB, -12 dB, -6 dB continuously observing control and response accelerometers for anomalies.
      ii. Full (0 dB) input test
   c. Generate control and response spectral plots.
   d. Verify base input levels were within random testing tolerances.
   e. Visually inspect flexure and mirror for any physical damage, yield of components and backed out fasteners.

8. Post-Random Sine Sweep test
   a. Follow procedure step 6 to assess if fundamental frequencies changed during random input test.

9. Overlay and compare pre and post random vibration sine sweep test plots. Verify the difference is within specified limits.
10. Remove vibes fixture and place in the autocollimator setup to inspect any deviation in the mirror from before the test.

Test Procedures for Y axis
12. Repeats steps 2 through 10 for Y axis

Test Procedures for Z axis
13. Remove vibes fixture and attached to the vertical plate adapter to complete Z axis testing on the horizontal shake table.
14. Repeat steps 2 through 10 for Z axis testing.