

Design and Analysis of Satellite Subsystem Supporting Structure

Ranganath N¹, Dr. Panchakshari H V², Dr. A Ramesh³, Hareesh A⁴

Mechanical Department, KSIT, Bengaluru, Mechanical Department, RRIT, BITS Institute of Technology, Hindupur, Department of Mechanical Engg, KSSEM

Abstract: In the era of scientific and technological advancements continuous improvements of the existing systems is becoming the order of the day. More emphasis is given towards the R&D and adoption of latest trends. The application of Aluminum materials in the field of space technology has proved the performance improvements in terms of reduced mass and increased stiffness. Design and Analysis of satellite support system using aluminium is taken up for study. The main objective is to design a support structure for meeting the given constraints of geometry and stiffness with minimum mass. Various analyses were performed to select the final configuration. Free vibration analysis was done to assess interaction between the subsystem and supporting structure. Sensitivity analyses were performed by varying the design parameters of the structure. Linear static analysis was performed to study the response of structure for static loads. Buckling analysis was performed to study the critical loads of support structure. The design analysis is carried out using Design and Analysis software I-DEAS.

Keywords: Sub system, Vibration, Buckling, Static, FE model, Analysis

1. Introduction

Increased accuracy of payloads, the on-orbit structural dynamic behavior of spacecraft is increasingly influencing the design and performance of spacecraft. Present work is on the design of a satellite subsystem support structure assembly for mass reduction with safe operating conditions. The forces imparted to the spacecraft are important for sensitive instruments. In this report a safe configuration is suggested with significant achievement in mass reduction. Normal mode analysis and sensitivity analysis were performed to select an optimized design configuration. The requirements of such a support structure are high stiffness, low weight and high strength.

The structure is connected to the deck of the satellite and it carries subsystems mounted on it. The top surface of the bracket is connected to the deck of the satellite.

As materials used for the spacecraft application must be lightweight and also retain their stiffness throughout the spacecraft mission in orbit. Subsystem support structure is basically designed for stiffness requirements. While keeping stiffness as the design criteria, the structure will be checked to withstand static and dynamic loads arising during launch phase.

Specification of subsystem considered for Design:

- Sub system mass and Inertia.
- Mass : 1kg
- $I_{xx} = 5384.81 \text{kg}\cdot\text{mm}^2$, $I_{yy} = 4661.9 \text{kg}\cdot\text{mm}^2$, $I_{zz} = 1004.72 \text{kg}\cdot\text{mm}^2$ C.G location = 20mm above the subsystem interface.

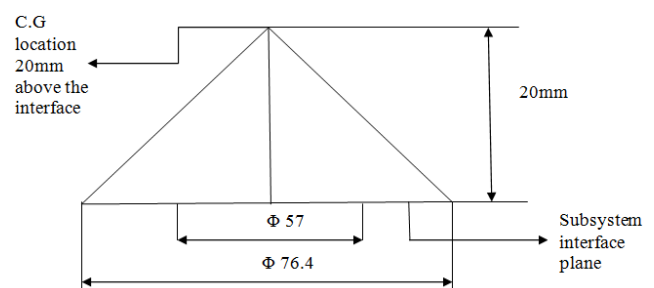


Figure 1: Representation of sub system

- First natural frequency > 70 Hz: To avoid coupling between bracket and rest of the spacecraft.
- Static analysis: 20g load taken in X, Y and Z directions independently.
- Buckling analysis: 20g load taken in X, Y and Z directions independently.

2. Problem Description

The main objective of the structural design is to achieve the minimum mass structure, which will satisfy the stiffness and strength requirements. Hence, optimum configuration, newer technological achievements have to be incorporated to attain minimum mass, which at the same time satisfy all the basic requirements.

Material Properties

Material used for subsystem support structure is AA-2024. Whose properties are given below:

Material	E (N/m ²)	ν	G (N/m ²)	Density(kg/m ³)
Aluminium	70E+9	0.30	26.8E+9	2800

E = Young's Modulus of the Aluminum

ν = Poison's Ratio

G = shear Modulus of the Aluminum

3.Steps in Analysis

The following types of analysis are carried out:

1. Free Vibration Analysis
2. Static Analysis
3. Buckling analysis

1. Free vibration analysis (Eigenvalue analysis)

For a structural system with a total DOF of N, the stiffness matrix **K** and mass matrix **M** have a dimension of N ×N. In this technique, first solve the homogenous equation. The homogeneous equation is by considering the case of **F** = 0, therefore it is also called free vibration analysis, as the system is free of external forces. For a solid or structure that undergoes a free vibration, the discretized system equation becomes.

$$\mathbf{K}\mathbf{D} + \mathbf{M}\mathbf{D}'' = 0 \text{ (Eq.2.1)}$$

This solution for the free vibration problem can be assumed as

$$\mathbf{D} = \boldsymbol{\varphi} \exp(i\omega t) \text{ (Eq.2.2)}$$

Where $\boldsymbol{\varphi}$ is the amplitude of the nodal displacement, ω is the frequency of the free vibration, and t is the time.

By substituting Eq. (2.2) into Eq. (2.1), we obtain

$$[\mathbf{K} - \omega^2\mathbf{M}] \boldsymbol{\varphi} = 0 \text{ (Eq.2.3)}$$

or

$$[\mathbf{K} - \lambda\mathbf{M}] \boldsymbol{\varphi} = 0 \text{ (Eq.2.4)}$$

where

$$\lambda = \omega^2 \text{ (Eq.2.5)}$$

Equation (2.3) (or (2.4)) is called the eigenvalues equation. To have a non-zero solution for $\boldsymbol{\varphi}$, the determinate of the matrix must vanish:

$$\det [\mathbf{K} - \lambda\mathbf{M}] = |\mathbf{K} - \lambda\mathbf{M}| = 0 \text{ (Eq.2.6)}$$

The expansion of the above equation will lead to a polynomial of λ of order N. This polynomial equation will have N roots, $\lambda_1, \lambda_2, \dots, \lambda_N$, called eigenvalues, which relate to the natural frequency of the system by Eq. (2.3).

4.Model Description

Subsystem support bracket (Option 1)

The size of bracket is 210mm x 270mm with height of 200mm. the top surface of the bracket consists of six interfaces used for connecting the bracket with deck. There are two cutouts dia.57mm, around which four interfaces each are provided to fix the subsystem.

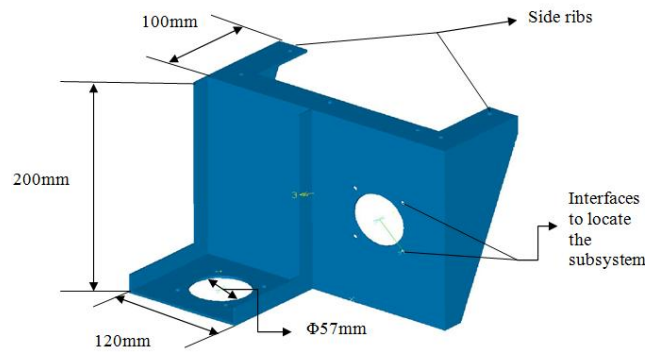


Figure 2: 3D model

1. FE model:

Finite element model is shown in figure. Four noded quadrilateral shell elements having 5-degree of freedom per node is used to model the bracket. Mass of the subsystem is defined at center of gravity location. Lumped mass element each of 1kg is attached at the C.G location points to simulate the subsystem mass. This mass is equally distributed on the insert location of the subsystem by means of multipoint constraints (MPC).

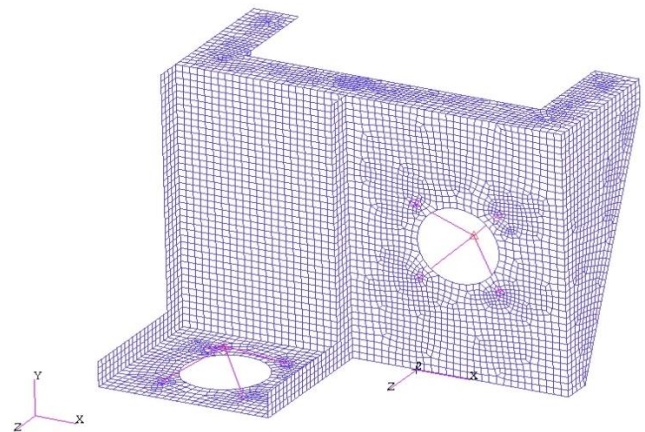


Figure 3: FE model

2. Boundary conditions:

For free vibration analysis, the boundary condition set consists of only the restraints set. The entire node at the insert location of the bracket is restrained as Tx, Ty, Tz, Rx, Ry and Rz are fixed.

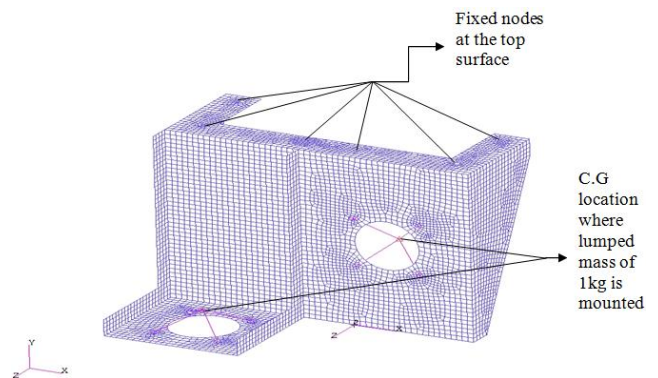
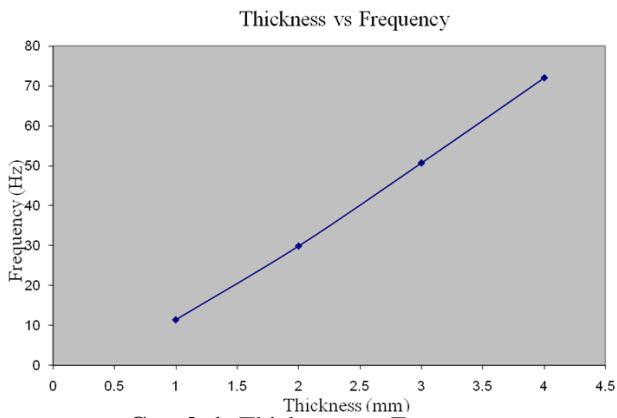


Figure 4: Boundary condition at top surface

3. Design parametric study:

The design parameter like thickness of the model was varied in order to select a suitable configuration. The variation of frequency with change in thickness for the model of configuration 1 is as shown below:



Graph 1: Thickness vs. Frequency

As it is observed from the graph 1 the change in the thickness of the bracket results in the change of the natural frequency of the structure. Since, the required frequency of 70 Hz is achieved with 4mm thicknesses that results in 1.2kg mass of the bracket. So, in order to minimize the mass for the desired frequency, design modification in the bracket is required. The mode shape of first and second natural frequencies with 2mm shell thickness is shown in fig below.

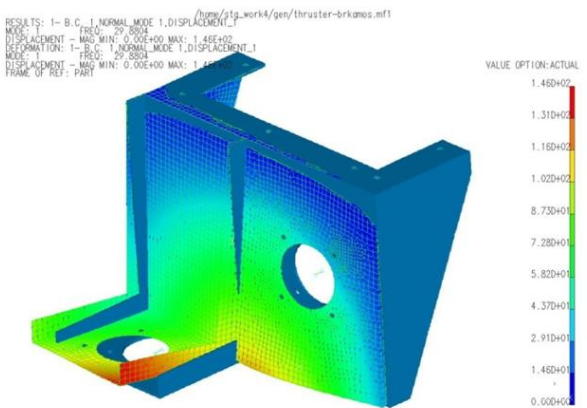


Figure 5: First mode shape (29.88Hz)

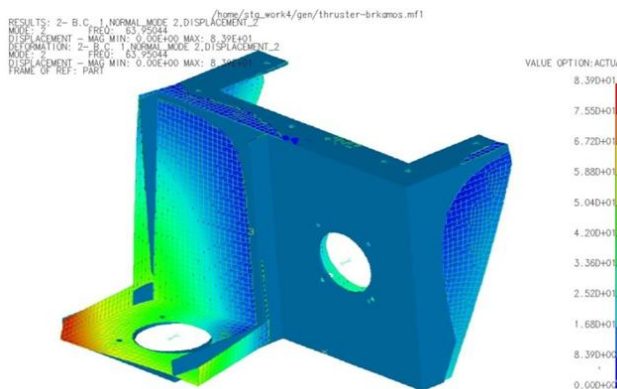


Figure 6: Second mode shape (63.95Hz)

Modified Subsystem Support Bracket (Option 2)

In the modified bracket one additional rib is provided at the mid with suitable interfaces. All other major dimensions are same as previous bracket.

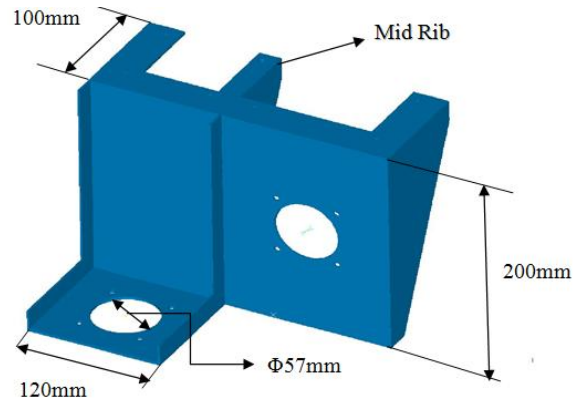


Figure 7: 3D model

1. FE model:

Finite element model is shown in figure 8. Four noded quadrilateral shell elements having 5- degree of freedom per node is used to model the bracket. Mass of the subsystem is defined at C.G location. Lumped mass element each of 1kg is attached at the C.G location points to simulate the subsystem mass. This mass is equally distributed on the insert location of the subsystem by means of multipoint constraints (MPC).

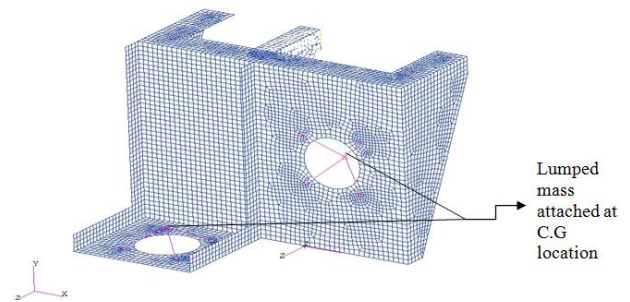


Figure 8: FE model

2. Boundary condition:

For free vibration analysis the boundary condition set consists of only the restraints set.

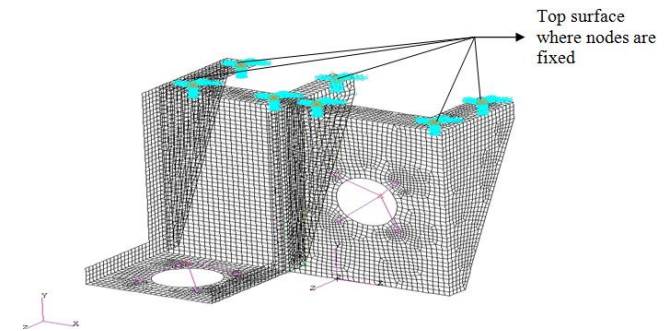
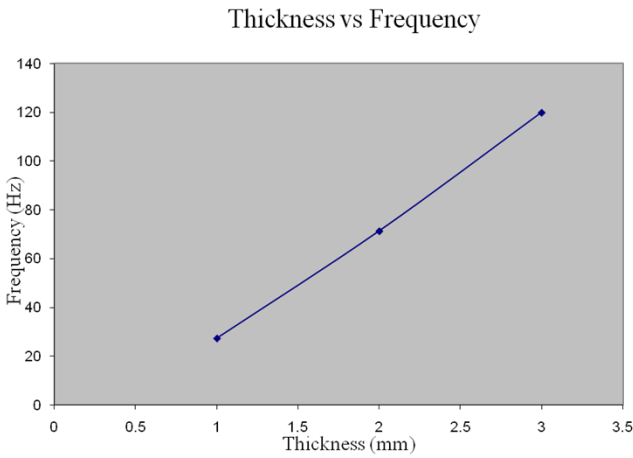


Figure 9: Boundary conditions at top surface

4. Design parametric Study:

The design parameter like thickness of the model was varied in order to select a suitable configuration. The variation of frequency with change in thickness for the designed thruster bracket is as shown below:



Graph 2: Thickness vs Frequency

As it is observed from the graph that change in the thickness results in the change of the natural frequency of the structure. The required frequency of 70 Hz is achieved with 2 mm thickness resulting mass of 0.697kg against 1.2kg as compared to the previous bracket. So, this model satisfies the stiffness criterion with minimum mass.

5. Static Analysis

In case of static structural analysis the loads are assumed to be applied slowly and the model is constrained to prevent rigid body motion. That is, Static equilibrium exists for the model.

Model Description:

Linear static analysis was carried out to compute the response of structure for 20g load in X, Y and Z directions independently. I-DEAS linear static analysis was used for required purpose. The load entry was used to define the direction and magnitude of a gravity vector in Global Coordinate System.

6. Results and Discussion

Von mises stresses and interface forces are calculated for 20g load in X, Y, Z directions independently and results are given in table 4.1 and 4.2

Table 1: von mises stresses

load	Von mises stress (MPa)	Factor of safety (FOS)	Margin of safety (MOS=FOS-1)
20g-X direction	212	1.27	0.27
20g-Y direction	138	1.95	0.95
20g-z direction	212	1.27	0.27

Table 2: Check for interface forces

Load	Force in Kgf
20g-X direction	18.8
20g-Y direction	10.6
20g-Z direction	36.2

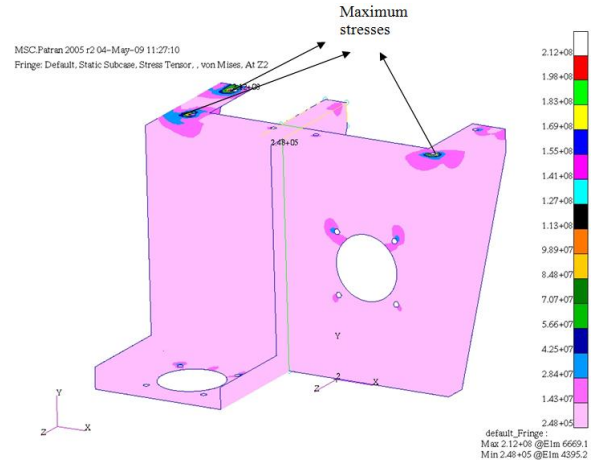


Figure 10: von mises stress for 20g load in X direction

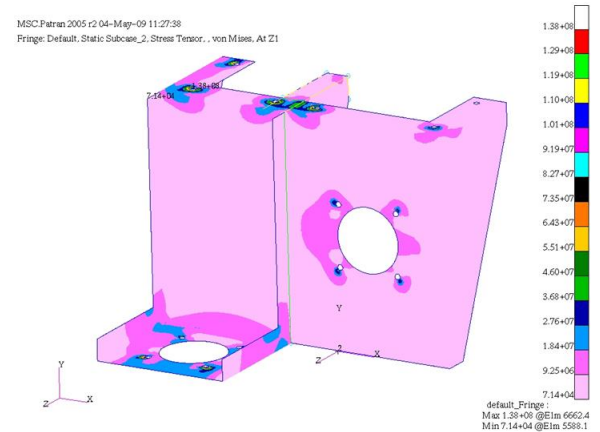


Figure 11: von mises stress for 20g load in Y direction

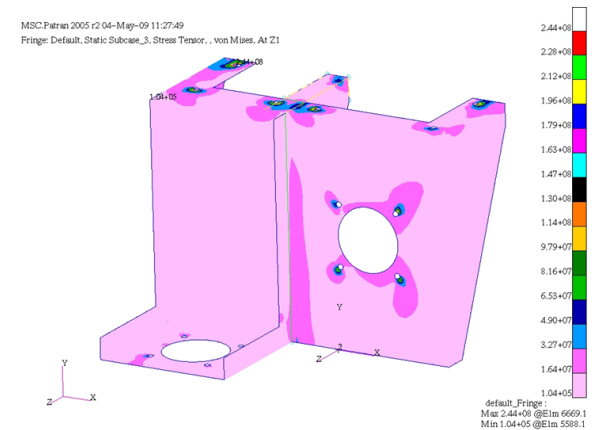


Figure 12: von mises stress for 20g load in Z direction

7. Buckling Analysis

In linear static analysis a structure is normally considered to be in a state of stable equilibriums the load is removed, the structure is assumed to return to its original position. However, under certain combination of loading, the structure may become unstable. When this loading is reached, the

structure continues to deflect without increase in magnitude of loading. In this case, the structure has actually buckled or has become unstable. Present study is performed for 20g forces in X, Y and Z direction.

Model Description:

Linear buckling analysis was carried out to compute the buckling failure of the structure for 20g force in X, Y and Z directions respectively. The fixed boundary condition was assumed at the connecting holes on the top surface of the bracket. I-DEAS Linear buckling analysis was used for the required purpose.

8.Results and Discussion

Buckling Load (20g)	Buckling Factor
Along X	23.48
Along Y	47.32
Along Z	11.22

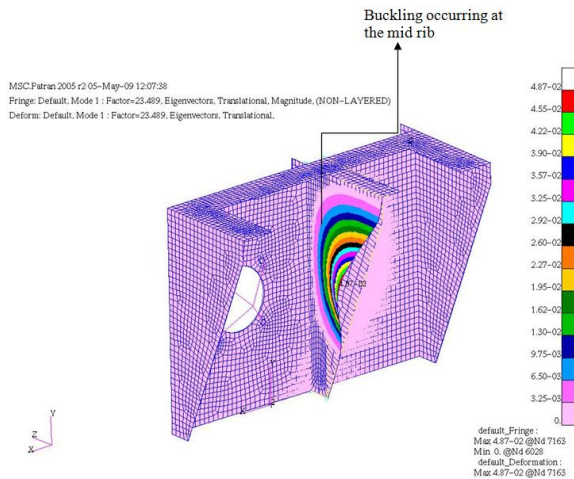


Figure 13: Buckling load for 20g in X direction

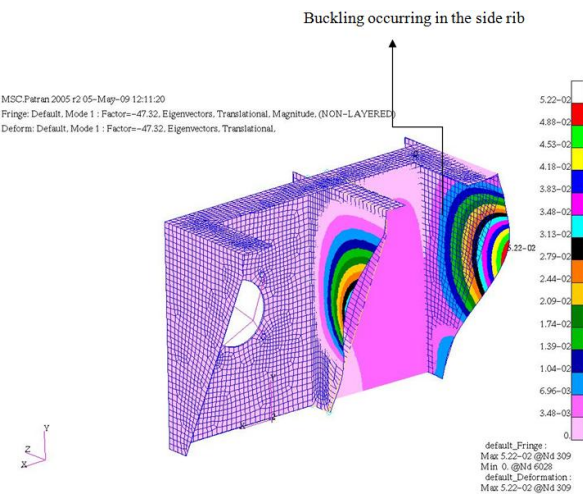


Figure 14: Buckling load of 20g in Y direction

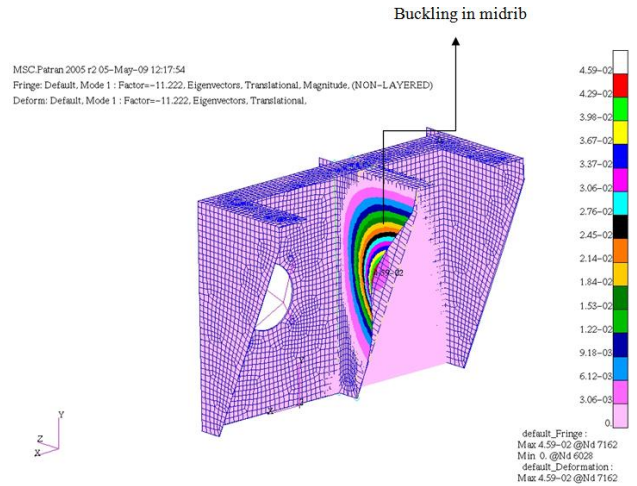


Figure 15: Buckling load of 20g in Z direction

9. Conclusion

The satellite subsystem support structure has been designed to support the deck of the satellite. The sensitivity analysis was carried out by varying design parameter like thickness. The configuration studied satisfies the stiffness requirement. The table below shows the comparison between Option 1 and Option 2.

SL.NO	Parameters	Option 1	Option 2
1	Thickness (mm)	4	2
2	1 st mode Frequency (Hz)	71.99	71.29
3	2 nd mode frequency (Hz)	148.8	84.48
4	Mass (kg)	1.2	0.697

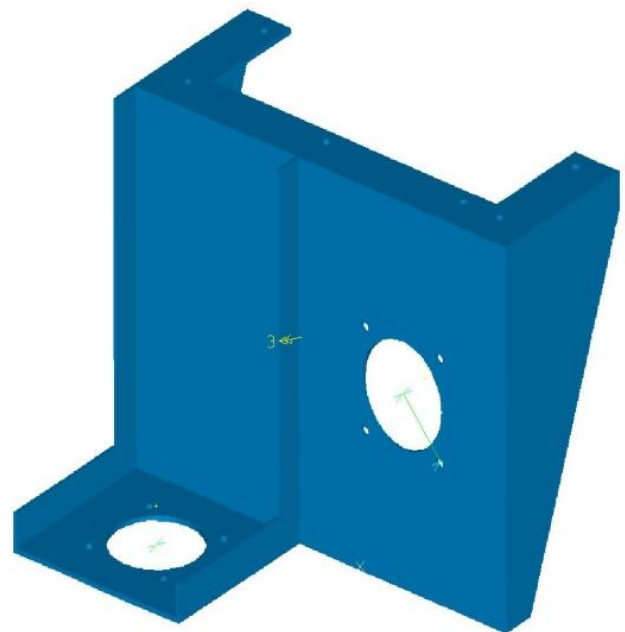


Figure 16: Option 1

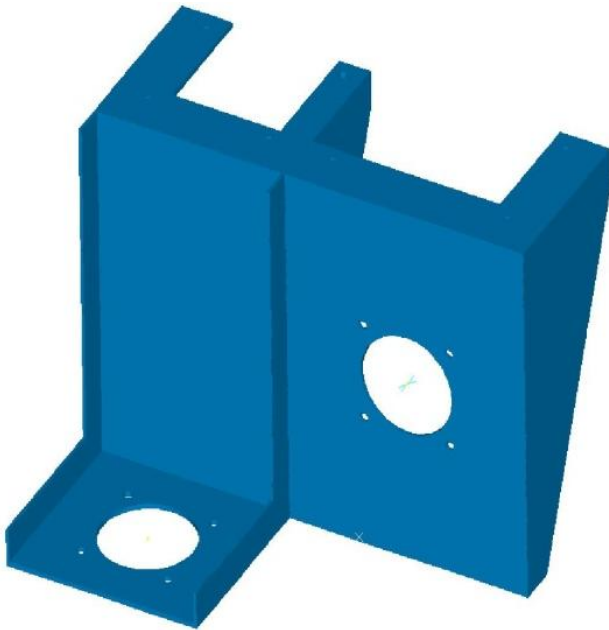


Figure 17: Option 2

- [15] Kerslake, W. R., Byers, D. C., and Staggs, J. F., "SERT II: Mission and Experiments," *Journal of Spacecraft and Rockets*, Vol. 1, No. 1, 1970, pp. 4 – 6
- [16] Nakanishi, S., and Pawlik, E. V., "Experimental Investigation of a 1.5-m-Diam Kaufman Thruster," *Journal of Spacecraft and Rockets*, Vol. 5, No. 7, 1968, pp. 801 – 807

References

- [1] The Elmer A. Sperry Award 2004
- [2] Roll-Royce manual.
- [3] A. Meerkerk, S. Zaalberg, N. Wit, W. V. Wijk. H. Kievit, *Hybrid Propulsion-Electrical Installations*, Rotterdam Mainport University of Applied Sciences-RMU.
- [4] Thrustmaster corporate brochure 2013
- [5] I. Martinez, *Marine Propulsion*.
- [6] *Ship terms and definitions*.
- [7] C.C. Liang, W.H. Cheng, *The Optimum Control of Thruster System For Dynamically Positioned Vessels*, Science Direct, *Ocean Engineering* 31 (2004) 97–110.
- [8] D. Barton, *Failure Modes and Effect Analysis of Diving Support Vessel-Adams Challenge*.
- [9] C. Dymarski, M. Zagórski, *New Design of the Poded Azimuth Thruster for a Diesel-Hydraulic Propulsion System of a Small Vessel*, *Journal of Polish CMAC*, 2011.
- [10] M. Fonte, L. Reis, M. Freitas, *Failure Analysis of A Gear Wheel Of A Marine Azimuth Thruster*, *Engineering Failure Analysis*, October 2011
- [11] S. Go, Y. Ahn, J. Kim, H. Seo, *A Study on the Powering Performance of Drillship in Transit Mode with Azimuth Thrusters*, *Third International Symposium on Marine Propulsors smp'13*, Launceston, Tasmania, Australia, May 2013.
- [12] T. Kim, S. Lim, H. Seok, C. Kang, *Concurrent Engineering Solution for the Design of Ship and Offshore Bracket Parts and Fabrication Process*, *International Journal of Naval Architecture and Ocean Eng.* (2013) 5:376–39.
- [13] A.V. Parab, *Steering Torque Determination of Pulling Open Thrusters*, Delft University of Technology, Jan 2015.
- [14] Islam M. F., Akinturk A, Liu P., Veitch B., *Performance Characteristics of a Podded Propulsor During Dynamic Azimuthing*, 8th Canadian Marine Hy.