Modal analysis of Support bracket for air compressor system

Lokesh[1], Pavan Prabhakar[2]

[1]Post Graduate student, Department of Mechanical Engineering, New Horizon college of Engineering, Bangalore, India
[2]Assistant professor, Department of Mechanical Engineering, New Horizon college of Engineering, Bangalore, India

---

Abstract - *This paper describes the static and modal Analysis of supporting bracket for air compressor system. Supporting bracket is a simple rigid structure in the shape of an L, one arm of which is fixed to vertical surface, the other projecting horizontally to support a shelf or other weight. Natural frequency of supporting bracket is calculated by performing a modal analysis using the finite element method. Common name for this type of analysis is finite element Analysis and failure of the supporting bracket is mainly due to resonance is quite significant in many industries, power plant, machineries etc, which leads to the serious problems in production, efficiency of the system and also financial losses. Hence performing modal analysis of those system is unavoidable and important, it is also one of the remedies to overcome those problems. This analysis provides the frequency and mode shape for various vibrating system which helps in improving performance and avoid damage to system and its surrounding. The modal analysis of the supporting bracket is performed using ABAQUS workbench is used as a FEA tool to find the natural frequencies of the supporting bracket.*

Keywords: supporting bracket; free vibration; finite element Analysis, Abaqus.

1. INTRODUCTION

In mechanical engineering a bracket is any intermediate component for fixing one part to another, usually larger part. What makes a bracket a bracket is the fact that it is intermediate between the two and fixes the one to the other. Brackets vary wildly in shape, but a prototypical bracket would be the L-shaped metal piece that attaches a shelf (the smaller component) to a wall (the larger component). a vertical arm mounted on the wall, and a horizontal arm projecting outwards for another element to be attached on top of it or below it. To enable the outstretched arm to support a greater weight, a bracket will often have a third arm running diagonally between the horizontal and vertical arms, or indeed the bracket may be a solid triangle. By extension almost any object that performs this function of attaching one part to another (usually larger) component is also called a bracket, even though it may not be obviously L-shaped. In our case the bracket is used for to supporting the compressor system mainly used the railways. This bracket is subjected to the dynamic load so modal analysis is carried out.

1.1 OBJECTIVE FORMULATION

Design for strength with operating requirement of the equipment.
Modal analysis to find the natural frequency and mode shapes for the supporting bracket of air compressor system
Analyze the vibration characteristic of supporting structure for air compressor

1.2 METHODOLOGY

- Creating the supporting bracket of air compressor model by CATIA V5
- Mesh Generation by HYPERMESH
- Exporting to ABAQUS
- Applying loads/boundary conditions
- Static analysis due to self weight
- Finding mode shape & Natural frequency
- Visualization
- Validation of Results

1.3 CALCULATIONS

Material properties

Material	= ASTM a 36 steel
Young's modulus	= 200 x 10^3 N/mm^2
Density	= 7.85x 10^{-9} tonne/mm^3
Poisons ratio	= 0.26
Yield strength	= 250N/mm^2
Factor of safety	= 1.5
Total mass	= 35.72 kg

Calculation for static analysis

In order to solve the solve this problem, consider the support bracket of air compressor system as a simple cantilever beam and load of the air compressor assembly is applied to cantilever beam.

Formula for calculation

Bending moment

M_b = force x perpendicular distance

Moment of inertia $I = (bd^3)/12$

Bending stress

$$\sigma_b = M_b\, y/I$$

Calculation for modal analysis

Deflection of the cantilever beam section [2]

$$\delta = \frac{Wa^3}{3EI} + \frac{Wa^2}{2EI}(L-a)$$

Natural frequency is given calculated by

$$fn = 1/2\Pi \; x\sqrt{(g / \delta)}$$

1.4 FE MODELING

The model is meshed using 2D shell element, Mesh element size of 5 mm is chosen.

Total Elements : 6661

Total Nodes : 6881

Boundary condition and loads

The 4 bolt location in the vertical arm fixed in all degree of freedom, C.G point of the motor has taken and motor mass of 6 kg is applied to that point, Rigid body element is used on bolt location to create a rigid region. Solis bolt element are not required, it uses a RBE element to represent bolt, as shown in Fig. 1.2. Tensile and bending loads can be transferred through the RBE nodes.

In the Air compressor supporting bracket motor load is transferred to the support bracket by using rigid element.

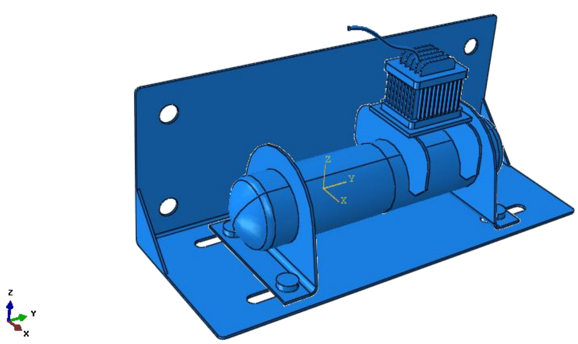

Fig-1: Cad Model

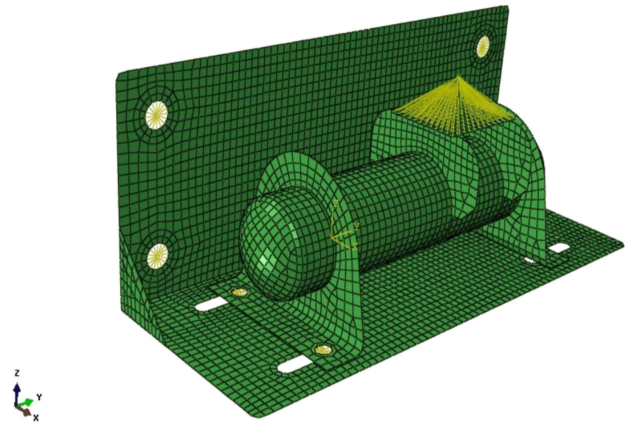

Fig-2: Meshed Model

1.5 FEM ANALYSIS

CASE-1 Static analysis due to self weight

Static analysis of the air compressor supporting bracket is carried out due to self weight.

FEM RESULT

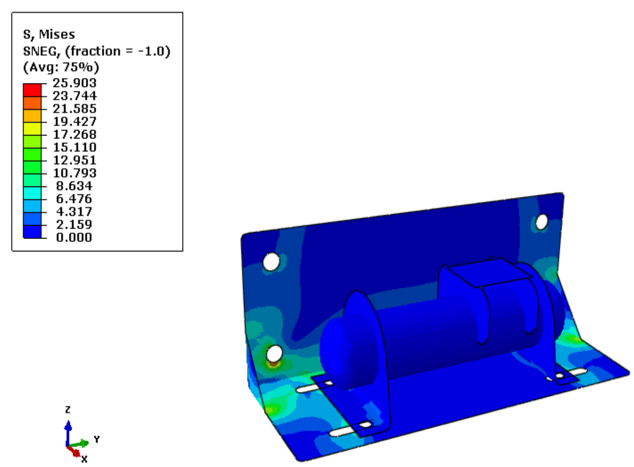

Fig-3: Static Analysis Result

The maximum stress obtained by FEM method is 25.90 N/mm^2

The maximum stress obtained by the analytical method is 26.64 N/mm^2..

Comparing with the permissible stress of 167N/mm^2, the calculated value 26.64 N/mm^2 is less than the permissible stress. Hence the bracket is safe under self load condition.

CASE-2 Modal analysis of support bracket

Mode shape & Natural frequency

A mode shape is a specific pattern of vibration executed by a mechanical system at a specific frequency. Different mode shapes will be associated with different frequencies. The Analysis technique of modal analysis discovers these mode shapes and the frequency

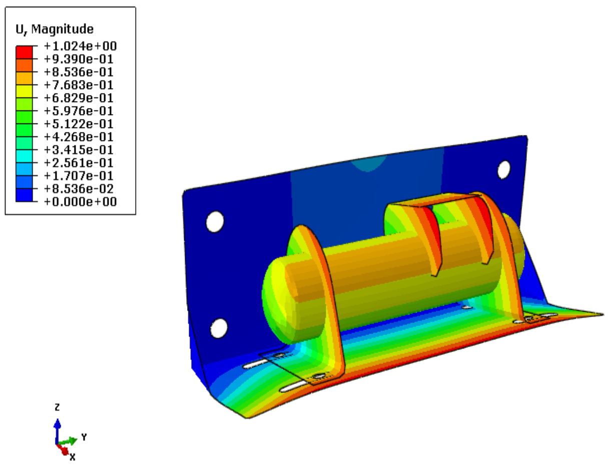

Fig-4: Mode1 at frequency 24.45 cycle/sec

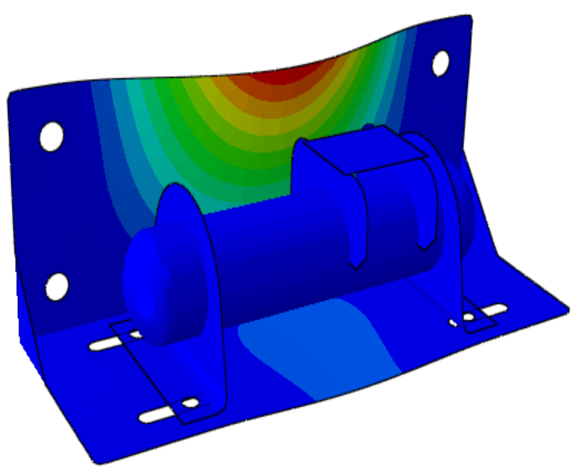

Fig-5: Mode2 at frequency 123.10 cycle/ sec

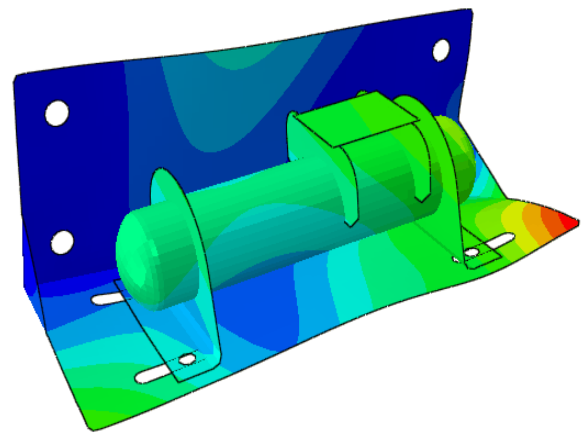

Fig-6: Mode3 at frequency 144.10 cycle/ sec

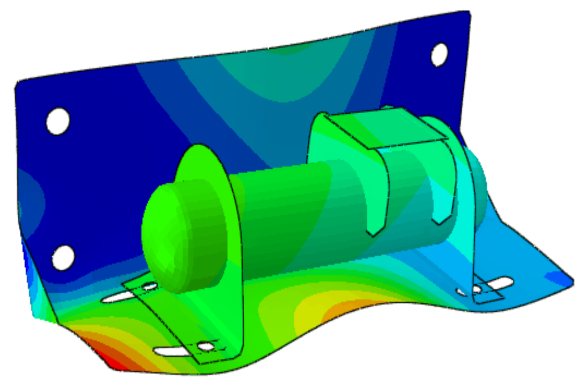

Fig-7: Mode4 at frequency 157.49 cycle/ sec

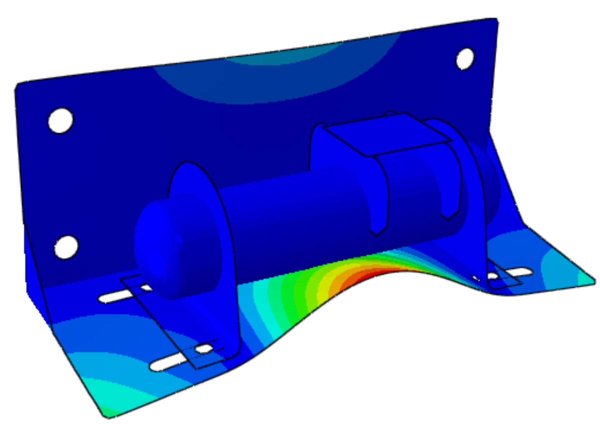

Fig-8: Mode5 at frequency 204.85 cycle/ sec

The obtained natural frequency of the mode 1is 24.45 cycle/sec
Natural frequency obtained by the analytical method is 21.24 cycle /sec
By comparing the analytical and theoretical the obtained error is 13.12%

Different modes and corresponding frequency listed below

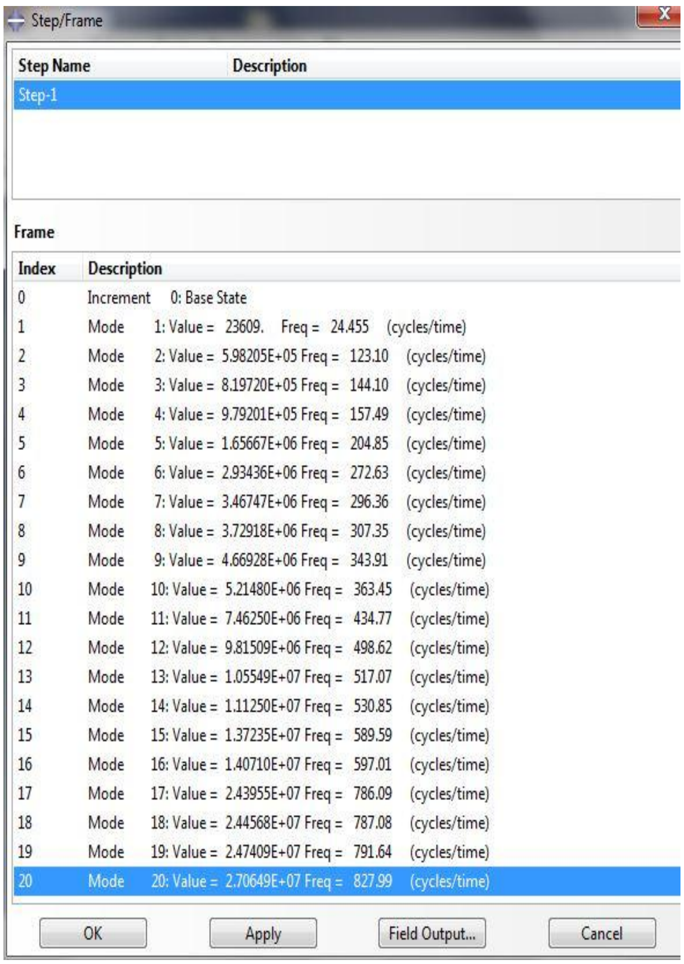

Fig-9: Different Modes and Frequency

3. RESULTS

In this project theoretical calculation is done by the above given formula by considering as cantilever beam and results are validated with the fem method

Table -1: Results of Support Bracket

	FEM method	Analytical method	% of variation
Case-1	Stress-25.90	26.64	2.77
Case-2	Fn-24.45	21.24	13.12

4. CONCLUSION

The strength criterion is satisfied with operating requirement of the equipment
The supporting bracket designed with stands static and dynamic, as per FEM analysis conducted and the design which is validated with the analytical.

REFRENCE

[1] Tarundeep singh, Mr dalgeet sing ”dynamic analysis of compressor mounting bracket ofautomobile air conditioning system” Feb 2000.

[2] Sangameshwar patil, prabhuling sarasambi, shravankumar b. kerur “dynamic analysis of composite l-shaped bracket” June 2005.

[3] Kaori fujita, isao sakamoto, yoshimitsu ohashi and masahiko kimura” static and dynamic loading tests of bracket complexes used in traditional timber structures in japan” June 2004.

[4] Monali Deshmukh, Prof. K R Sontakke” Analysis and Optimization of Engine Mounting Bracket” Dec. 2001.

[5] Tushar P. Kamble, Rajratna A. Bhalerao” “Optimization & Modal Analysis of Engine Mounting Bracket for Different Materials by Using Finite Element Analysis” August 2003.

[6] A.S. Adkine, Prof.G.P.Overikar and Prof. S .S. Surwase” static and modal analysis of engine supporting bracket - a literature review” December 2004.