MODELLING AND ANALYSIS OF STATIC AND DYNAMIC OF CONNECTING LUGS AND MECHANICAL BRACKET USING ABAQUS

M. VENKATESULU & CH. SAMBAIAH

Assistant professor, Department of Mechanical Engineering, Koneru Lakshmaiah Education
Foundation, Vaddeswaram, Guntur, Andhra Pradesh, India

ABSTRACT

This thesis contains the determination of static stress and displacement analysis of connecting lug of given load of 50 Mpa and the natural vibration analysis of mechanical bracket. Which has got a wide range of applications, such as connecting lugs are used in aero-heavy engineering industry to transfer the load from one component to another component. Connecting lugs are widely used to connect major aircraft components. Such joints are characterized by 100% load transfer through a pin or a bolt. This design offers several advantages over other solutions namely, simplicity, ease of assembly and free pivoting around the pin axis. A mechanical bracket for mounting a telecommunication antenna on a metallic tower comprises a pipe section to secure the bracket to the tower. The angular position of the pipe section about its longitudinal axis is selected to appropriately orient the bracket. A transversal pivot bridges a pair of spaced apart parallel plates of the bracket. The antenna comprises a pole member with a free end pivotally mounted on the pivot between the two plates. A bolt and nut assembly traverses the pole member and two accurate slots respectively made in the parallel plates, and hence the problem is selected.

Finite element method is used to obtain the solution. ABAQUS is a powerful finite element software package, which performs static and/or dynamic analysis and simulation of structures. It can deal with bodies with various loads, temperatures, contacts, impacts, and other environmental conditions. In case of static analysis, the stress and displacement are obtained for various step times, from 0.1 to 0.5 minutes with a time step of 0.1 minutes. In case of dynamic analysis, the frequency is obtained for various mode numbers, from 1 to 10 in steps of 1 mode number. The location at which the expected maximum and minimum of stresses and displacement values and frequency values are to be identifying.

KEYWORDS: Heavy Engineering Industry & Transversal Pivot Bridges

Received: Feb 22, 2018; **Accepted:** Mar 14, 2018; **Published:** Mar 30, 2018; **Paper Id.:** IJMPERDAPR2018144

INTRODUCTION

STATIC PROBLEM STATEMENT

The Connecting lug has got various applications in mechanical engineering and hence this problem is selected for static analysis.

Connecting lugs are used in aero-heavy engineering industry to transfer the load from one component to another component. Connecting lugs are widely used to connect major aircraft components. Such joints are characterized by 100% load transfer through a pin or a bolt. This design offers several advantages over other solutions namely, simplicity, ease of assembly and free pivoting around the pin axis.

PROBLEM DEFINITION

In this following sketch, we will use two-dimensional Connecting lug.

Where P, Load = 50Mpa

Figure: Connecting Lug

The Connecting lug is welded firmly to a massive structure at one end. The other end contains a hole. When it is in service, a bolt will be placed through the hole of the lug. We have been asked to determine the static stress and displacement of the connecting lug when a 50 Mpa load applied to the hole.

We will simplify this problem by making the following assumption:

- Neglect the variation of load around the circumference of the hole and use the uniform load. For the Connecting lug problem, we will perform the following tasks:
- Sketch the two-dimensional geometry and creating the part.
- Define the material properties.
- Assemble the model.
- Apply the loads and boundary conditions.
- Mesh.
- Creating a job and submit it for analysis.
- View the results of the analysis.
- Plot the stress vs time.
- Plot the displacement vs time.

MATERIAL

Steel

Material Properties

Young's modulus, $E = 200\text{Gpa}$

Poisson's ratio, $\nu = 0.3$

Load, $P = 50\text{ Mpa}$

The goal of this analysis is to examine the static response of the Connecting lug using ABAQUS/Standard.

DYNAMIC PROBLEM STATEMENT

The Mechanical bracket has got various applications in mechanical engineering and hence this problem is selected for static analysis.

A mechanical bracket for mounting a telecommunication antenna on a metallic tower comprises a pipe section to secure the bracket to the tower. The angular position of the pipe section about its longitudinal axis is selected to appropriately orient the bracket. A transversal pivot bridges a pair of spaced apart parallel plates of the bracket. The antenna comprises a pole member with a free end pivotally mounted on the pivot between the two plates. A bolt and nut assembly traverses the pole member and two accurate slots, respectively made in the parallel plate

PROBLEM DEFINITION

In this following sketch, we will use two-dimensional Mechanical Bracket.

Figure: Mechanical Bracket

The Mechanical Bracket is welded firmly to a massive structure at the left side of the top end. This problem contains natural vibration analysis of Mechanical Bracket. We have been asked to find the natural frequency at each model number of Bracket.

For this Mechanical Bracket, we will perform the following tasks:

- Sketch the two-dimensional geometry and creating the part.
- Define the material properties.
- Assemble the model.
- Apply the boundary conditions.
- Mesh.
- Creating a job and submit it for analysis.
- View the results of the analysis.
- Plot the natural frequency vs mode number.

MATERIAL

Aluminum.

Material Properties:

Young's modulus, $E = 70 \text{ Gpa}$.

Poisson's ratio, $\nu = 0.298$.

Density = 2700 kg/m^3 .

The goal of this analysis is to examine the dynamic response of the

Mechanical bracket using ABAQUS/Standard

RESULTS AND DISCUSSIONS

Visualization of the Results for Connecting Lug

In the **Module** list located below the toolbar, click **visualization** to enter the visualization module. We can see the minimum and maximum value of the **stresses** of the connecting lug:

Figure.1: Stress Results at Increment 1

Results in a figure.1, we found At Increment 1,

Set-up time = 0.1 min.

Maximum stress at node: 116 = 7.9999 N/mm^2 .

Minimum stress at node: 1 = 0.

Figure.2: Stress Result at Increment 2

Results in a figure.2, we found At Increment 2,

Set-up time = 0.2 min.

Maximum stress at node: 116 = 16 N/mm^2 .

Minimum stress at node: 1 = 0.

Figure.3: Stress Result at Increment 3

Results in figure.3, we found At Increment 3,

Set-up time = 0.3 min.

Maximum stress at node: 116 = 24 N/mm².

Minimum stress at node: 1 = 0.

Figure.4: Stress Result at Increment 4

Results in figure.4, we found At Increment 4,

Set-up time = 0.4 min.

Maximum stress at node: 116 = 32 N/mm².

Minimum stress at node: 1 = 0.

Figure.5: Stress Result at Increment 5

Results in figure.5, we found At Increment 5,

Set-up time = 0.5 min.

Maximum stress at node: 116 = 39.9999 N/mm².

Minimum stress at node: 1 = 0.

A static analysis is performed on the connecting lug loaded with 50 mpa. The step time varies from 0.1 to 0.5 minutes, in steps of 0.1 min and to get the minimum and maximum values of stresses is obtained. In this connecting lug analysis results, we found the maximum stress at increment 5 = 39.9999 N/mm².

Why maximum stress values are changing as change “increment” number, despite the fact that no change in “node” number is observed.

The analysis has carried is linear static analysis. The ABAQUS software solver incrementally to converge the results. So the load is gradually increased in this case and since load direction is not changing the same location the high stress are observed in different increment.

Table: Plot the Graph Between Stress Vs Time:

Time(min)	Stress(N/mm ²)
0.1	7.9999
0.2	16
0.3	24
0.4	32
0.5	39.9999

Graph

Need for Drawing the Graph

This graph showing clearly between stress and time. In tabular columns not possible to see the variation between these stresses and time. This graphical representation indicates where the stress is maximum and minimum.

It is observed that at lower time steps the stress is lower, when the step time is increased automatically the stress is also increased. In this graph, we observed the stress is directly proportional to step time. The stress is minimum (i.e. 7.9999n/mm²) at step time 0.1minute and stress is maximum (i.e. 39.9999n/mm²) at step time 0.5minutes.

In the visualization of the result, we can see the minimum and maximum values of the **displacements** of the connecting lug:

Figure.6: Displacement Result at Increment 1

Results in a figure.6, we found at Increment 1,

Set-up time = 0.1 min.

Maximum displacement=0.000582mm.

Minimum displacement=0.

Figure.7: Displacement Result at Increment 2

Results in figure.7, we found At Increment 2,

Step time = 0.2 min.

Maximum displacement=0.001163mm.

Minimum displacement=0.

Figure.8: Displacement Result at Increment 3

Results in figure.8, we found At Increment 3,

Step time = 0.3 min.

Maximum displacement=0.001745mm.

Minimum displacement=0.

Figure.9: Displacement Result at Increment 4

Results in figure.9, we found At Increment 4,

Step time = 0.4 min.

Maximum displacement=0.002327mm.

Minimum displacement=0.

Figure.10: Displacement Result at Increment 5

Results in figure.10, we found At Increment 5,

Step time = 0.5 min.

Maximum displacement=0.002909mm.

Minimum displacement=0.

A static analysis is performed on the connecting lug loaded with 50 mpa. The step time is varied from 0.1 to 0.5 minutes in steps of 0.1 min and to get minimum and maximum values of displacement is obtained.

In this connecting lug analysis results we found the maximum displacement at increment 5 = 0.002909mm.

Table: Plot the Graph Between Displacement Vs Time

Time(min)	Displacement(mm)
0.1	0.000582
0.2	0.001163
0.3	0.001745
0.4	0.002327
0.5	0.002909

Need for Drawing the Graph

This graph showing clearly between displacement and time. In tabular columns not possible to see the variation between these displacements and time. This graphical representation indicates where the displacement is maximum and minimum.

When the step time is increased automatically and the displacement is also increased. In this graph, we observed the displacement is directly proportional to step time. The displacement is minimum (i.e. 0.000582mm) at step time 0.1 minute and displacement is maximum (i.e. 0.002909mm) at step time 0.5minutes.

Visualization of the Results for Mechanical Bracket

In the **Module** list located under the toolbar, click **visualization** to enter the visualization module. We can see the value of frequency at each modelnumber of the mechanical bracket. In this mechanical bracket problem, we take 15 mode numbers.

Figure.11: Frequency Results at Mode Number 1

Results in figure.11, we found at modelnumber 1,

Frequency = 1453.6 cycles/time.

Figure.12: Frequency Results at Mode Number 2

Result in figure.12, we found at modelnumber 2,

Frequency = 4198.4 cycles/time.

Figure.13: Frequency Results at Mode Number 3

Results in figure.13, we found at modelnumber 3,

Frequency = 11823 cycles/times.

Figure.14: Frequency Results at Mode Number 4

Results in figure.14, we found at modelnumber 4,

Frequency = 18142 cycles/time.

Figure.15: Frequency Results at Mode Number 5

Results in figure.15, we found at modenumber 5,

Frequency = 21865 cycles/time.

Figure.16: Frequency Results at Mode Number 6

Results in figure.16, we found at modenumber 6,

Frequency=26212 cycles/time.

Figure.17: Frequency Results at Mode Number 7

Results in figure.17, we found at modenumber 7,

Frequency = 32697 cycles/time.

Figure.18: Frequency Results at Mode Number 8

Results in figure.18, we found at modenumber 8,

Frequency = 35639.

Figure.19: Frequency Results at Mode Number 9

Results in figure.19, we found at modenumber 9,

Frequency = 36623 cycles/time.

Figure.20: Frequency Results at Mode Number 10

Results in figure.20, we found at modenumber 10,

Frequency = 44611 cycles/time.

The dynamic analysis is performed on the mechanical bracket, the model number is varied from 1 to 10 in steps of 1 and to get the minimum and maximum values of frequency is obtained.

In this mechanical bracket analysis result, we found the maximum frequency is obtained at the modelnumber 10 = 44611 cycles/time.

Table: Plot the Graph between Frequency Vs Model Number

Model Number	Frequency(Cycles/Time)
1	1453.6
2	4198.4
3	11823
4	18142
5	21865
6	26212
7	32697
8	35639
9	36623
10	44611

Need For Drawing the Graph

This graph is used for graphical representation of the entire system, which is used to knowing unskilled persons also clearly understanding where the frequency maximum and minimum at mode numbers. Maximum frequency (i.e. 44611cycles/time) is obtained at mode number 10, minimum frequency (1453.6 cycles/time) is obtained at mode number 1.

When the mode number is increased automatically the frequency is increased slowly, at lower mode number the frequency is lesser and high mode number the frequency is higher.

CONCLUSIONS

In this work determined the static stress and displacement analysis of connecting lug of given load of 50 Mpa and the natural vibration analysis of mechanical bracket.

It is dealt with bodies with various loads, temperatures, contacts, impacts, and other environmental conditions. In case of static analysis, the stress and displacement is obtained for various step times, from 0.1 to 0.5 minutes with a time step of 0.1 minute. In case of dynamic analysis, the frequency is obtained for various mode numbers, from 1 to 10 in steps of 1 mode number. The location at which the maximum and minimum of stress values, displacement values and frequency values are identified which is clearly explained in the above said result and discussions.

REFERENCES

1. <https://academy.3ds.com/en/software/abaqus-student-edition>
2. https://en.wikipedia.org/wiki/Static_program_analysis
3. <http://searchsoftwarequality.techtarget.com/definition/dynamic-analysis>
4. <https://www.veracode.com/blog/2013/12/static-testing-vs-dynamic-testing>
5. <https://www.youtube.com/watch?v=uBYpBTFd98M>
6. <http://bobcat.nus.edu.sg:2080/English/SIMACAEGSARefMap/simagsa-c-matmodelexa.htm>