ANALYSIS & DESIGN OF TUBULAR TESTRIG

Nusra S[1], Anu A[2]

[1]PG Scholar, Structural Engineering, YCET, Kollam, Kerala, India
[2] Assistant Professor, Dept. of Civil Engineering, YCET, Kollam, Kerala, India

---

Abstract - *This paper presents the analysis and design of tubular steel frame testrig for the qualification testing of semicryo actuator used in the launch vehicle. The modal analysis was performed for the test rig structure with the engine and actuator, so as to achieve the suitable safety criteria in terms of structural frequency for the rig structure compared to actuator testing frequency. The engine and actuator supporting members are designed manually and are analyzed by Finite Element method using ANSYS software.*

Key Words: Testrig, ANSYS, Modal analysis, Actuator, Frequency, Finite Element Analysis, Tubular sections, Circular hollow sections

1. INTRODUCTION

Thrust vectoring, or thrust vector control or TVC, is the ability of a launch vehicle to manipulate the direction of the thrust from its engine in order to control the attitude or angular velocity of the launch vehicle. A test rig is a large structure used for the testing of the engine actuators used in the launch vehicles for thrust vectoring. Few of the parameters of actuators tested in the rig are frequency response, step response and flight profile. The test rig consists of a steel framed structure designed to support the engine and actuator. More than one actuator can be tested at one time. The test rig consists of a frame assembly and a pit to accommodate the dummy engine and actuator. The dummy engine is assembled to the rig so as to produce the same mass and moment of inertia as that of the actual engine. The design of the frame structure for the test rig is the crucial aspect in order to ensure that structural frequency of rig is sufficiently higher than the frequencies up to which the actuator will be tested.

1.1 Tubular sections

Tubular steel components are gaining popularity in the construction of steel framed structure. This is mainly because greater strength-to-weight ratios than open sections, greater design properties in both axis such as moment of inertia, section modulus and radius of gyration and high resistance to torsional loading. Many examples demonstrate the excellent properties of the circular hollow section as a structural element in resisting compression, tension, bending and torsion. Further, the circular hollow section has proved to be the best shape for elements subjected to wind, water or wave loading. The circular hollow section combines these characteristics with an architecturally attractive shape. Structures made of hollow sections have a smaller surface area than comparable structures of open sections. This in combination with the absence of sharp corners, results in better corrosion protection. These excellent properties result in light designs with a small number of simple joints in which gussets or stiffening plates can often be eliminated. Although the unit material cost of hollow sections is higher than that of open sections, this can be compensated by the lower weight of the construction, smaller painting area for corrosion protection and reduction of fabrication cost by the application of simple joints without stiffening elements.

1.2 Scope

This study is scoped to few limitations as followed:

- The analysis and design will be restricted for the frame structure only
- Analysis carried out using computer software ANSYS
- Analysis and design will be conducted according to the Indian Standard specifications and other available relevant standards

1.3 Methodology

- Carry out literature study to find out the objectives of the project work
- Understand the design procedure of various members
- Analyze all the models using Finite element analysis by ANSYS software
- Evaluate the analysis results

2. THE FINITE ELEMENT METHOD

The basis of FEA relies on the decomposition of the domain into a finite number of sub-domains called elements for which the systematic approximate solution is constructed by applying the variational or weighted residual method. FEA reduces the problem to that of a finite number of unknowns by dividing the domain into elements and by expressing the unknown field variable in terms of the assumed approximating functions within each elements. These functions are defined in terms of the values of the field

variables at specific points referred to as nodes. Nodes are usually located along the element boundary and they connect adjacent elements.
The ability to discretize the irregular domains with finite elements make the method a valuable and practical analysis tool for the solution of boundary, initial and eigen value problems arising in various engineering disciplines
The finite element analysis method requires the following major steps

- Discretization of the domain
- Selection of interpolation functions
- Development of element matrix for the sub-domain
- Assembly of the element matrices
- Solution of equations
- Computation of derived quantities

2.1 Dynamic Analysis

Dynamic analysis is concerned with the behaviour of the continuum under prescribed boundary conditions and dynamically applied loads; dynamic loads are applied as a function of time. This time-varying load application induces time-varying response (displacements, velocities, accelerations, forces, and stresses). These time-varying characteristics make dynamic analysis more complicated and more realistic than static analysis.
The usual first step in performing a dynamic analysis is determining the natural frequencies and mode shapes of the structure with damping neglected. These results characterize the basic dynamic behaviour of the structure and are an indication of how the structure will respond to dynamic loading. The natural frequencies of a structure are the frequencies at which the structure naturally tends to vibrate if it is subjected to a disturbance. The deformed shape of the structure at a specific natural frequency of vibration is termed its normal mode of vibration.

2.2 Modal analysis

Modal analysis is the study of natural characteristics of structures. The response of structures is different at each of the different natural frequencies. These deformation patterns are called mode shapes. Natural frequency and mode shapes are used to design structural systems for noise and vibration applications. The results of modal analysis are obtained as Eigen values and Eigen vectors. The Eigen values represent the modal frequencies while the Eigen vectors give the modal shapes of the various vibration modes. Generally the first mode is of greater interest as it has the maximum energy associated with it. Dynamic analysis involves loads and responses that vary with time. Modal analysis is a method to describe a structure in terms of its natural characteristics which are the frequency, damping and mode shapes. Modes are inherent properties of a structure, and are determined by the material properties and boundary conditions of the structure. Each mode is defined by a natural frequency, modal damping and a mode shape. If either the material properties or the boundary conditions of a structure change, its modes will change.
Normal mode analysis determines the natural frequencies and mode shapes for a structure. The natural frequencies of a structure are the frequencies at which the structure will naturally tend to vibrate if subjected to a disturbance. For example, the strings of a piano are each tuned to vibrate at a specific frequency. Other terms for natural frequency include characteristic frequency, fundamental frequency, resonance frequency, and normal frequency. Each mode shape is associated with a specific natural frequency.

2.3 Performing Finite Element analysis using ANSYS

The general steps that can be used for setting up any finite element analysis in ANSYS is follows,

- Defining the Problem
- Specifying the Type of Analysis
- Creating the Model Geometry
- Defining the Finite Elements
- Representing Boundary Conditions
- Specifying Material Properties
- Applying the Loads and controlling Analysis output
- ANSYS Output and reviewing the results

2.4 ANSYS Preprocessor

Model generation is conducted in this processor, which involves material definition, creation of a solid model, and, finally, meshing. Important tasks within this processor are:

- Specify element type.
- Define real constants (if required by the element type).
- Define material properties,
- Create the model geometry.
- Generate the mesh.

2.5 ANSYS Solution Processor

This processor is used for obtaining the solution for the finite element model that is generated within the Preprocessor. In the solution level, the loading conditions such as point load or pressure and constraints or boundary conditions are specified and finally the resulting setof equations are solved. Loads and boundary conditions can be applied in both the Preprocessor and the Solution processor.

- Define analysis type and analysis options,
- Specify boundary conditions.
- Obtain solution.

2.6 ANSYS General Postprocessor

In this processor, the results at a specific time over the entire or a portion of the model are reviewed. The General Postprocessor is used to look at the results over the whole model at one point in time. This is the final objective of everything discussed so far; finding the stresses, deflections, temperature distributions, pressures, etc. These results can then be compared to some criteria to make an objective evaluation of the performance of our design. This stage provides different tools to view the results including:

- Lists of nodal displacements
- Element forces and moments
- Deflection plots
- Stress contour diagrams

3. MODAL ANALYSIS OF TESTRIG

Modal analysis is used to determine a structure's vibration characteristics such as natural frequencies and mode shapes. It is the most fundamental of all dynamic analysis types and is generally the starting point for other, more detailed dynamic analyses. The modal analysis is done in the test rig so as to achieve safety criteria in terms of frequency of test rig structure. The testing frequency of actuator is up to 10 Hz. So the structural frequency of the test rig should be higher than that of the actuator frequency.
Modal analysis in the ANSYS family of products is a linear analysis. Any non-linearity, such as plasticity and contact elements are ignored even if they are defined. We can select from among several mode-extraction methods: Block Lanczos, Super node, PCG Lanczos, un-symmetric, damped and QR damped. The damped and QR damped methods allow you to include damping in the structure. [2]
The general process for a modal analysis consists of these primary operations

- Build the model
- Apply loads and obtain solution
- Reviewing the results

3.1 Building the model for Modal analysis

For performing a modal analysis, only linear behavior is valid in. If nonlinear elements are specified, then ANSYS treats them as linear. Material properties defined in the analysis are Young's modulus and density. The material properties of various members used in the model is given in Table 1.

Table-1: Material properties

Material used	Mild Steel
Young's Modulus	2.1 x10^{11} N/m^2
Poisson's Ratio	0.3
Density	7850kg/m^3

The finite element model of the test rig consist of mainly beam elements to connect nodes. The engine assembly is directly joined with the rig at the bottom centre of the top section.
Total number of nodes = 308
Total number of elements = 928

3.2 Applying Loads and Obtaining the Solution

In this step, the analysis type and options are specified. Loads are applied and load step options are defined, and the finite element solution for the natural frequencies is started as follows:

- Enter the Solution Processor
- Define Analysis Type and Options
- Apply Loads
- Specify Load Step Options
- Solve
- Exit the Solution Processor

The numerical method for mode extraction is also specified in this step. Block Lanczos method is specified as mode extraction method. This is because

- Efficient extraction of large number of modes in most models
- Typically used in complex models with mixture of elements
- Efficient extraction of modes in a frequency range
- Handles rigid-body modes well

In a modal analysis, the only loads that affect the solution are displacement constraints. The constraints applied in this analysis was the column ends fixed for all the four columns.

3.3 Reviewing the Results

Results from a modal analysis consist of, natural frequencies, expanded mode shapes and relative stress and force distributions if needed.
For the modal analysis of test rig with engine assembly, the natural frequencies for the first ten mode shapes are requested, so as to compare the frequency of test rig with actuator testing frequency.

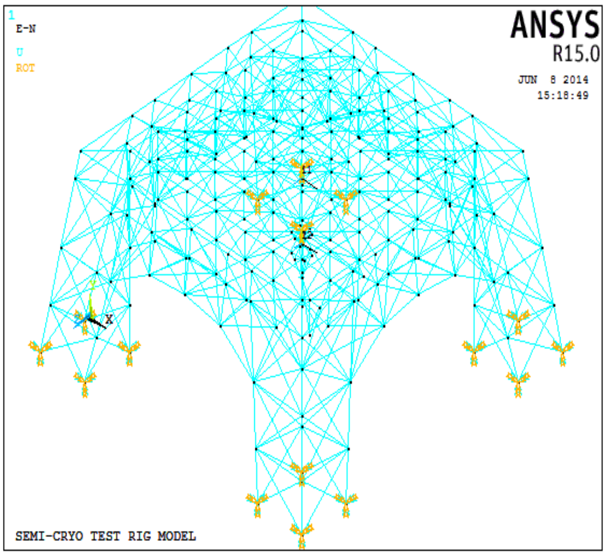

Fig-1: Finite element model of the test rig with engine assembly

The analysis was performed to extract first ten mode shapes of the test rig with frequency in each mode. The results of the model analysis of the test rig with engine assembly is discussed

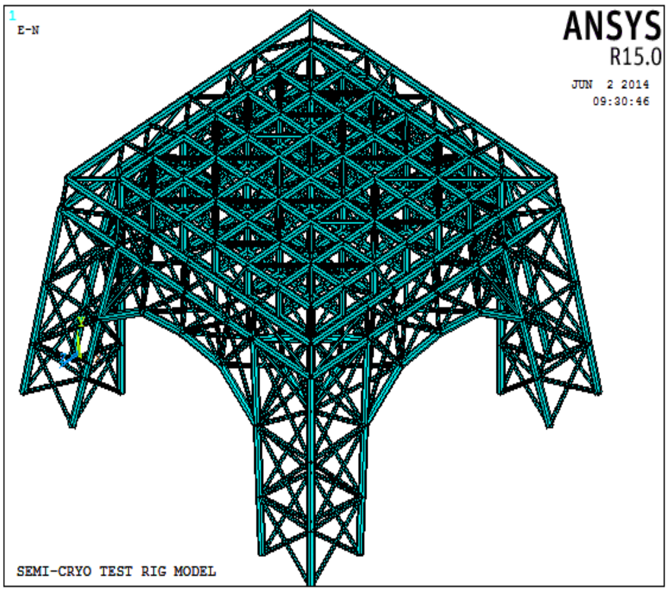

Fig-2: Existing configuration of test rig

4. RESULTS AND DISCUSSIONS

Modal analysis was performed for the test rig structure with engine assembly in ANSYS. First ten mode shapes with corresponding natural frequencies were obtained as results. The mode shapes extracted from the modal analysis of the test rig with engine assembly is shown in Fig. 3 to Fig.13

Table -2 : Results of modal analysis for the test rig

Sl. No.	Mode Number	Frequency (Hz)	Maximum displacement (mm)
1	1	10.066	0.02854
2	2	15.0413	0.05379
3	3	25.6479	0.046276
4	4	27.1378	0.032111
5	5	46.0263	0.047254
6	6	58.5142	0.036394
7	7	75.0249	0.006888
8	8	77.4659	0.0066968
9	9	82.6955	0.009944
10	10	90.921	0.006164

The maximum actuator testing frequency in the test rig is about 10 Hz. From the results of model analysis it is clear that the natural frequency for fundamental mode of the test rig is obtained as 10.066 Hz, which is greater than the actuator testing frequency. Also the maximum displacement for all the ten modes are less than 1 mm. Hence the structure is safe enough for the testing of engine actuators.

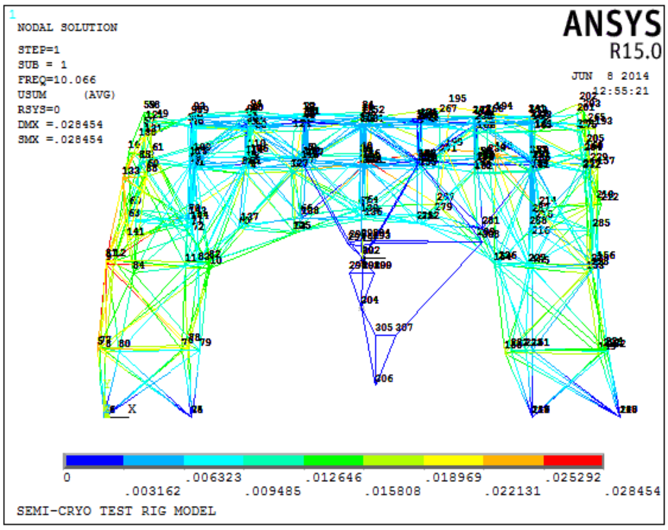

Fig-3: First mode shape with frequency 10.066 Hz

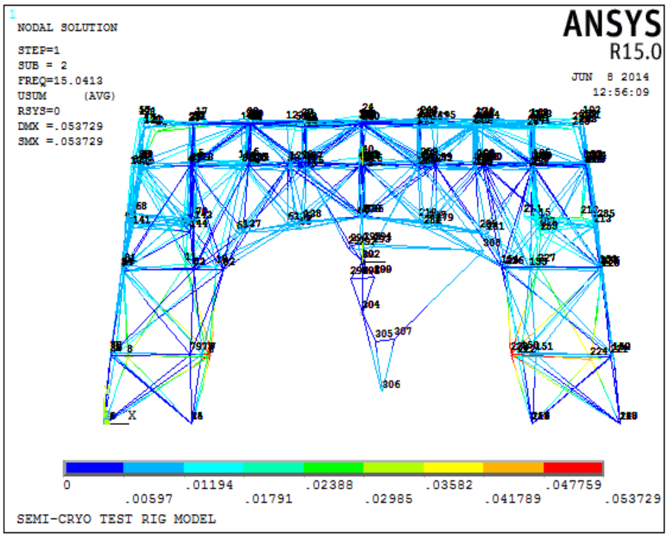

Fig -4: Second mode shape with frequency 15.0413 Hz

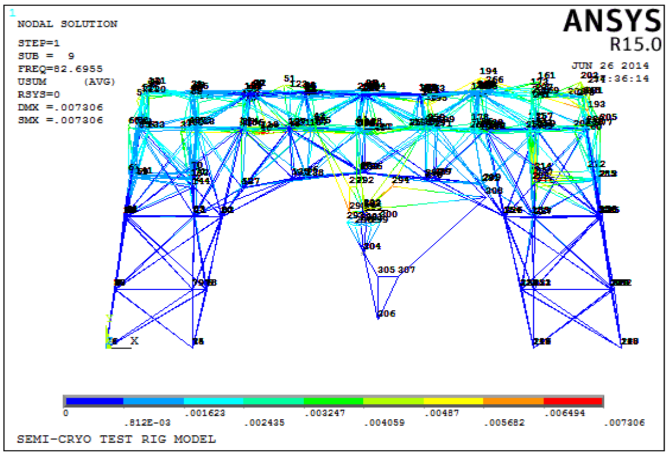

Fig -5 Ninth mode shape with frequency 82.6955 Hz

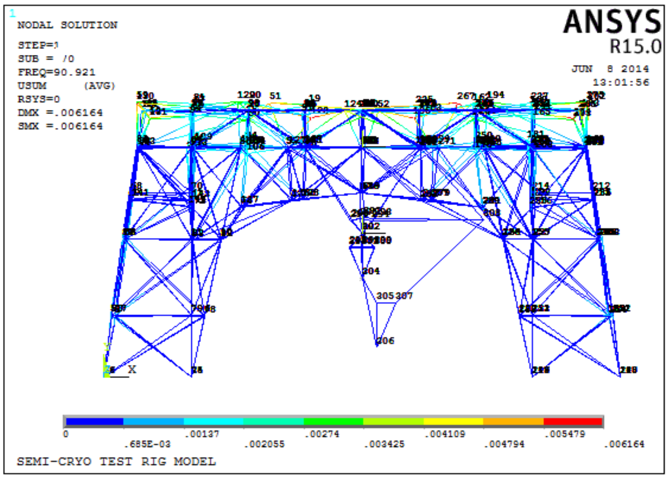

Fig -6 Tenth mode shape with frequency 90.921 Hz

5. CONCLUSIONS

Based on the present study reported following conclusions have been arrived at: Detailed design of the inertia simulated semicryo engine is carried out. The sizing of the members are done though hand computation and the final verification is done using the finite element software. Modal analysis of the test rig with the inertial simulated engine is carried out in order to ensure that the structural frequency of the rig - actuator combination is higher than the frequency up to which the actuator will be tested. The structural frequency of the test rig with engine assembly for the free un-damped model analysis is obtained for the fundamental mode is higher than the frequency up to which the actuator will be tested. Hence the design of test rig is adequate.

6. SUGGESTIONS FOR FUTURE WORKS

- The study can be extended by using other finite element soft wares such as ABAQUS, NASTRAN, etc.
- Attempts can be made to study behavior of test rig structure with other types of available standard steel sections.
- Attempts can be made to study experimentally, the behavior of test rig by small scale model studies.

REFERENCE

[1] B. Ganesh and Rahul Jhanwar,"Design of 25kW Redundant Linear Electro-Mechanical Actuator for Thrust Vector Control Applications,"Proceedings of the 1st International and 16th National Conference on Machines and Mechanisms, IIT Roorkee, India, Dec 18-20 -2013

[2] F.Adamcik and J. Labun"The Property Comparison of Electromechanical and Electro-hydraulic Flight Control Actuators," Advances in Military Technology, Vol. 5, No. 1, June 2010

[3] Ansys User's Manual, *Version 12,* Swanson Analysis Systems Incorporated, 2010

[4] CIDET: "Design guide for circular hollow section (CHS) joints under predominantly static loading" - 2nd edition 2008 (www.cidect.com

[5] M.C. Bittencourt and L.R. Lima (2007), "A Numerical Analysis of Tubular Joints under Static Loading, " APCOM'07 in conjunction with EPMESC XI, December 3-6, 2007, Kyoto, JAPAN

[6] IS 800-2007: "Indian Standard code of practice for general construction in steel"

[7] EN 1993-1-8 (2005*) (English*): Eurocode 3: Design of steel structures - Part 1-8: Design of joints

[8] IS 806- 1968, "Indian standard code of practice for use of steel tubes in general building construction"