International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2013): 6.14 | Impact Factor (2013): 4.438

Finite Element Analysis of Link Mechanism in Vertical Carousel Machine

Sammed Arinjay Patil[1], V. J. Khot[2]

[1]Mechanical Dertment Dr. J.J.Magdum College of Engineering Jaysingpur, Shivaji University Kolahapur

[2]Professor, Mechanical Dertment Dr. J.J.Magdum College of Engineering Jaysingpur, Shivaji University Kolahapur

Abstract: ***The purpose of FE Analysis is to model the behavior of the vertical carousel mechanism structure under a system of load. In order to perform the analysis all influencing factors must be considered and determined whether their effects are considerable or negligible on the final results. The degree of accuracy to which any system can be modeled is very much dependent on the level of planning that has been carried out. All the parameters and factors have to be accounted while planning the analysis. 'Planning an Analysis' is directly deals with the improving the results under specified condition. The analysis is carried out in three stages - Pre-Processing, Solution, Post Processing. Modal analysis is also done to obtain response of the mechanism under predetermined excitation. Software packages used for the analysis are CATIA V5, Hypermesh 9.0 and MSC NASTRAN.***

Keywords: Finite element analysis, vertical carousel machine, CATIA V5, MSC NASTRAN, Hypermesh 9.0

1. Introduction

This paper presents kinematic synthesis of linkage mechanism of vertical carousel machine. . Modeling of the guide-way and supporting frame is done on CATIA V5 software. It is then imported in Hypermesh 9.0 which is pre-processor software and simulated. Its finite element modeling is done. Finite element analysis is done using MSC NASTRAN as post processor. Optimization is carried out by using trial and error method. Three cases are analyzed for the given load conditions. Link thicknesses are taken 10mm, 8mm and 5mm for optimization. Plots deformation and equivalent stresses are obtained for each case. Modal analysis is also done in MSC NASTARN for three excitation frequencies. O. A. Bauchau et al. have presented multibody dynamics approach using finite element methods [1]. Sammed Patil et al. have presented synthesis of link mechanism in vertical carousel machine.[2]

However no work is performed regarding finite element analysis of vertical carousel mechanism. The modal analysis also can be used further for kinetic analysis. In order to do the finite element analysis, this paper presents a combination of CAD (Computer Aided Design) software, FEA (finite element analysis) software and graphical method is used [3].

2. System Under Investigation

The 3D view of the mechanism to be investigated is shown in figure 1 which is modeled in CATIA V5 software. As the figure indicates, the system is assembly of many components.

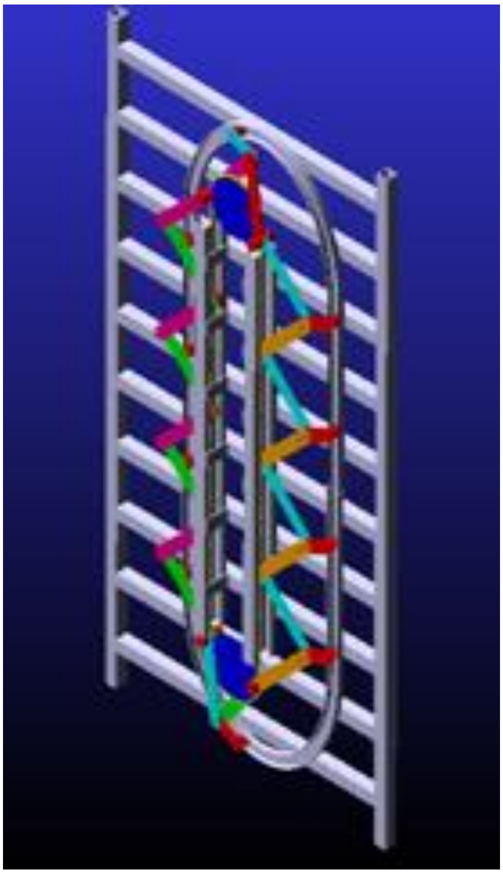


Fig. Half part of carousel machine mechanism

The description of components is expressed in table I, II, III and IV.

Table 1: Description of links

Body	Length mm	Width mm	Depth mm	Quantity
Main link	300	50	5	20
Guide link	100	50	5	10

Table 2: Description of roller and pin

Body	Length mm	Radius mm	Quantity
Roller	25	20	30
Outer Pin	50	2.5	10
Inner pin	80	2.5	20

Table 3: Description of guide-way

C channel	Length mm	Inner width mm	Outer width mm	Thickness mm	Quantity
Inner	1550	40	50	5	4
Outer	1427	40	50	5	2

Table 4: Description of elliptical guide-way

Half major axis mm	Half minor axis mm	Inner width mm	Outer width mm	Thickness mm
295	245	40	50	5

Table 5: A) Description of uniform chain

Chain pitch mm	Chain width mm	Pitch to back, mm	Roller diameter, Mm
10	10	3.4	6.5

Table 5: B) Description of sprocket

Type of sprocket	Sprocket width Mm	No. of teeth	Pitch diameter
Roller	8	78	266

3. Finite Elenet Modelnig

CAD model exported from CATIA V5 is imported in Hypermesh for further procedure. Initially geometry cleanup is performed to make the geometry free from all reference lines, points and plane used in modeling. Once the geometry cleanup operation is completed mid-surfacing is done for the parts to be meshed using shell (CQUAD4) elements. The guide ways and supporting frame is considered as a single united structure for the simplification of FE modeling, the structure formed by these two is meshed using second order tetra (CTETRA) elements. In order to get more accurate results, the frame is meshed with higher density. The main links and guiding links are meshed with shell elements. Guiding spheres are also modeled with second order tetra elements. Pins are modeled using bar (CBAR) elements in MSC Nastran. Pins are released at both ends in axial rotation to model the proper behavior, rigid (RBE2) elements are used to connect the pins with links. The refinement technique of meshing is used to mesh the model. The finite element model is having 347039 elements and 650299 nodes.

The mass which is to be placed in the tray is modeled using CONM2 element at their respective center of gravity. The mass elements are connected directly to the pins mounted between guiding links and main links. The sprockets and Chain are modeled using brick (CHEXA) elements. The sprockets and chain are simplified without modeling the teeth for time being. The contact between sprocket and chain is considered as bonded. Guiding spheres are connected to the links via pins which are modeled using bar elements. Advantage of symmetry is taken and symmetric model is used for analysis.

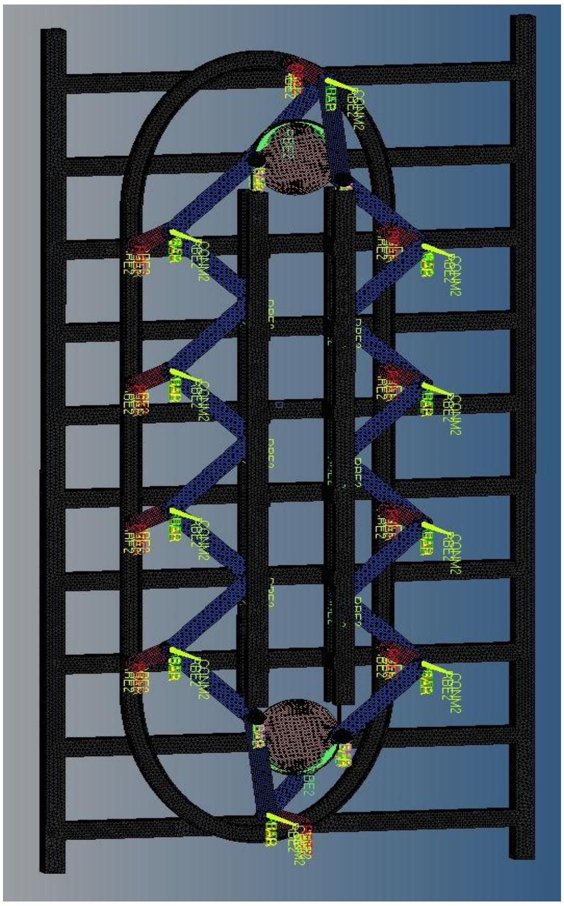


Figure: Shows Finite Element Model of Vertical Carousal Machine

Material Modeling and Load Conditions

Material properties have to be determined for the computer simulation. The loads are applied in the same units as the model geometry and material properties specified. The mass to be placed in the tray is modeled using the mass elements at their respective center of gravity and gravitational acceleration is defined to consider weight of those masses. Density of all parts has been defined to consider their self-weight in the analysis.

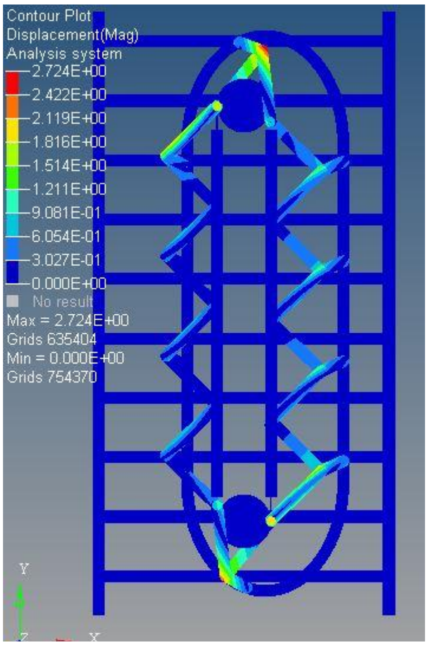
Property	*Young's Modulus (GPa)*	*Poisson's Ratio*	*Density (kg/mm^3)*
All Parts	210	0.3	7.860e-6
Rollers	23	0.35	1.20e-6

Boundary Conditions

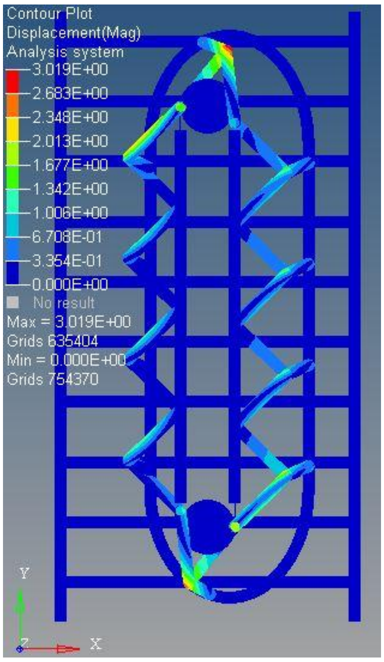
The force boundary conditions are imposed during the evaluation of the element matrices itself while the prescribed displacement boundary conditions are imposed after the assembly of the element matrices. Then the global system of linear equations is solved by any numerical technique to get the displacements at global nodes. The frame supporting the guide ways is constrained at the top and bottom end of the vertical side members using single point constraints (SPC's). The top and bottom ends are constrained in all directions using SPC.

Post Processing

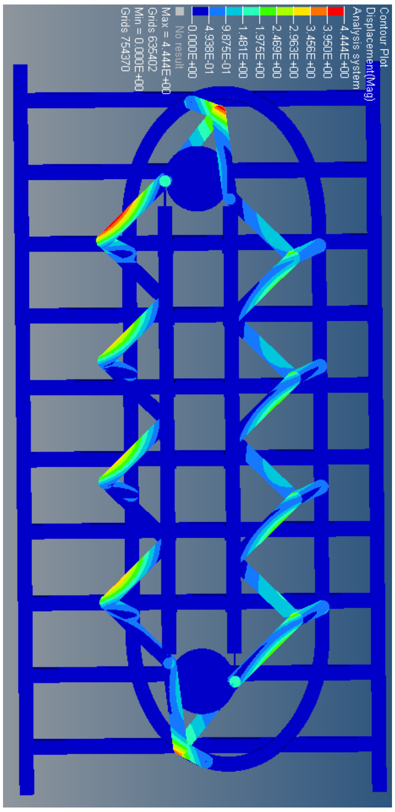
After completion of the analysis the result files are generated as per the preparation of the deck. The .OP2 file is generated which contains all the binary plots of the solution. This file containing all the results is imported in the post-processor (HYPERVIEW) for analyzing and plotting the results in graphical form. Displacement results can be displayed as a plot of the deformed element mesh superimposed over a plot of the unreformed model. Displacements can be scaled such that the deformed shape is exaggerated for clarity. For two-dimensional solid models, stress components can be displayed as contour plots. The stress components available from the solution are the normal, shear, and von Mises stress components for plane stress and plane strain, and the radial, axial, shear, and hoop stress components for axisymmetric models. For models using truss or beam elements, stress components are plotted as bar charts. Deformation plots for three cases are as follows



Case I (Link Thickness 10mm)

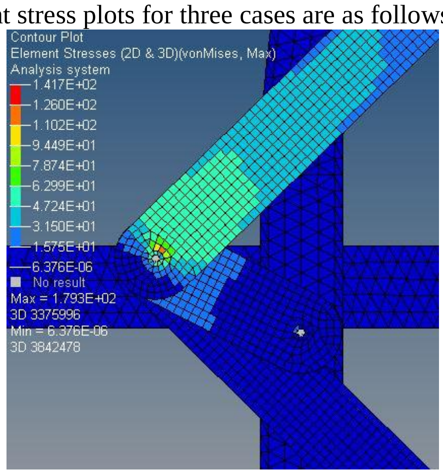


Case II (Link Thickness 8mm)

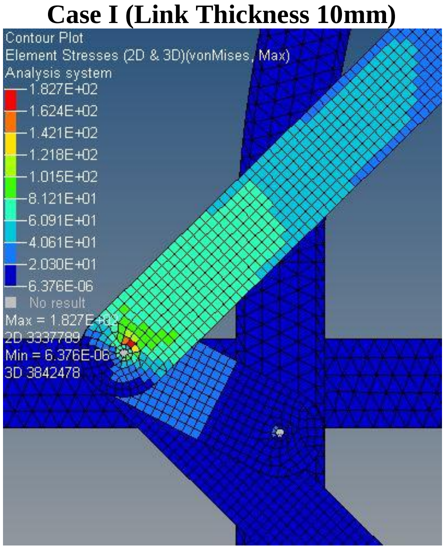


Case III (Link Thickness 5mm)

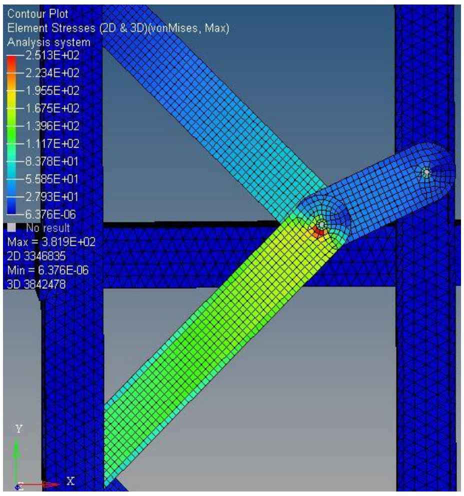
Equivalent stress plots for three cases are as follows



Case I (Link Thickness 10mm)



Case II (Link Thickness 8mm)



Case III (Link Thickness 5mm)

Modal Analysis

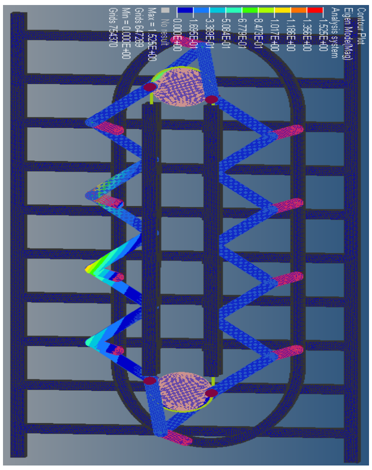
Modal analysis is important in machines where there is likely to be cyclic out of balance forces, such as in rotating machinery (engines, electric pneumatic motors, generators, industrial equipment etc.) and fluid flow applications (due to alternating vortex shedding). The chief aim of ant vibration analysis is to ensure that the system is not subject to dangerous resonant condition during the range of operation

Mesh Requirements for Eigen Value Analysis

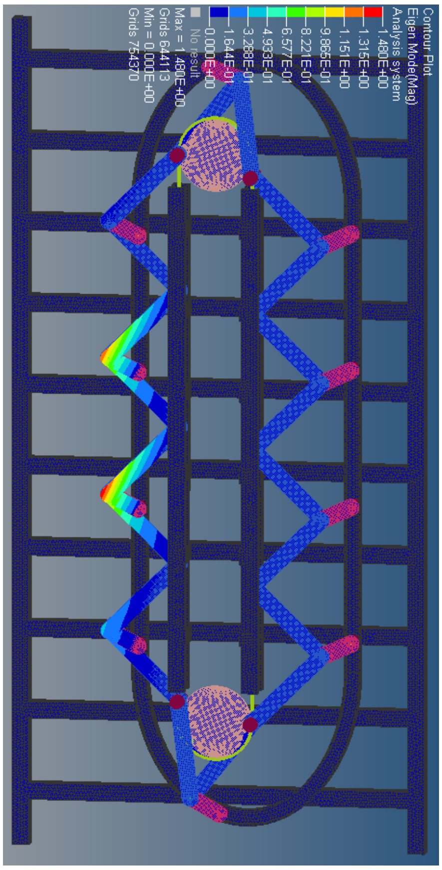
Since Eigen value problems are less sensitive to mesh density compared to other analysis types, coarse meshes can be used for the 3D model, so long as the mesh is graded and refined towards nodes which carry loads and boundary conditions. One should not be using overly coarse meshes, as it will result in a stiffer structure with resulting higher modes of vibration than is actually the case.

4. Results-Modal Analysis

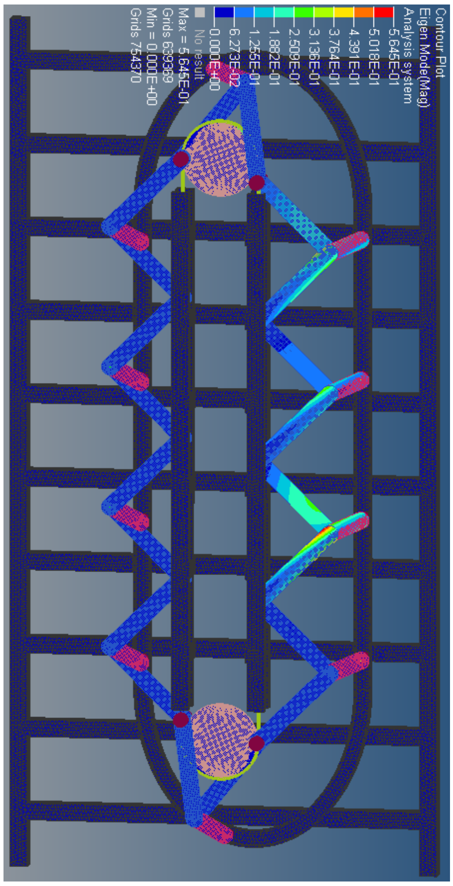
The main excitation source in the Vertical Carousal Machine is driving motor. The speed of driving motor ranges from 30 to 50 RPM, so the excitation frequency from 0.5Hz to 0.83 Hz. The first global mode in modal analysis is observed at the frequency of 3.03Hz which is too far from the operating frequency. Hence the Vertical carousal machine is safe against dynamic loading. First four global modes occurred during the analysis are shown below. The deformed shape is highlighted in color.



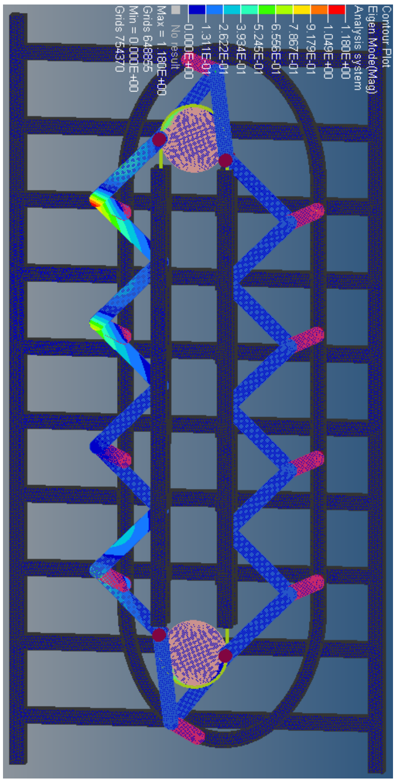
First Mode Shape (Frequency=3.03Hz)



Second Mode Shape (Frequency=3.06Hz)



Third Mode Shape (Frequency=3.33Hz)



Fourth Mode Shape (Frequency=4.52Hz)

5. Discussions

Maximum equivalent stress in pins in finite element analysis for each case is as follows

CASE No. (Thickness in mm)	Maximum equivalent stress (N/mm^2)
Case1 (10mm)	141.7
Case 2 (8mm)	182.7
Case 3 (5mm)	251.3

Maximum deformation of links in finite element analysis for each case is as follows

CASE No. (Thickness in mm)	Maximum deformation (mm)
Case1 (10mm)	2.72
Case 2 (8mm)	3.01
Case 3 (5mm)	4.44

For optimization on the basis of minimum is a best criterion, for selection of link thickness values of maximum equivalent stress and deformation for Case 3 are safe and acceptable by the client.

6. Conclusions

CAD modeling of link mechanism in vertical carousel machine in CATIA V5 and Finite element analysis as well as modal analysis in MSC NASTRAN can be successfully done. Further simulation and analysis with experimental setup is recommended.

7. Acknowledgements

The author would like to thank Prof. V. J. Khot, Department of Mechanical engineering, of Dr. JJMCOE, Shivaji University-Kolahpur and Mr. Sachin Patil of Tech-Mech Industries, Pune, Mr. Vikram Vasekar, Pune for their guidance during the project.

References

[1] O. A. Bauchau C. L. Bottasso Y. G. Nikishkov, "Modeling Rotorcrafk Dynamics with Finite Element Multibody Procedures", Mathematical and Computer Modelling, 33 PP:1113-1137, 2001.

[2] Sammed Patil, Prof. V. J. Khot. "Synthesis of Link Mechanism In Vertical Carousel Machine" [Patil, 4(6): June, 2015] ISSN: 2277-9655

[3] NASTRAN Software, MSC Softwares, http;//www.mscsofware.com/products/CAE-Tools/NASTRAN.aspx

Author Profile

Mr. Sammed A. Patil is M. E. –Research Fellow in Department of Mechanical Engineering, Dr. J J Magdum College of Engg, Jaysingpur, Maharashtra.

Prof. Vijay J Khot is Associate Professor at Dept of Mechanical Engineering, Dr. J J Magdum College of Engg, Jaysingpur, Maharashtra.