MULTIBODY DYNAMIC ANALYSIS OF MECHANISM FOR HEALTH EXCESCISE VIBRATOR

Rajashekar M[1], Ashok[2], Anita[3]

[1]*Assistant Professor, Department of Mechanical Engineering, BKIT, Bhalki-585328, Karnataka, India*
[2]*Project Group, Department of Mechanical Engineering, BKIT, Bhalki-585328, Karnataka, India*
[3]*Project Group, Department of Mechanical Engineering, BKIT, Bhalki-585328, Karnataka, India*

Abstract

In this paper health exercise vibrator mechanism is designed for specific performance of exercise. The three dimensional conceptual model has developed first in CATIA V5. Later kinematics motions were verified using CATIA and also using ADAMS software. By using the ADAMS, the dynamic parameters of three-body vibrating mechanism are analyzed and the simple finite element model is established on the basis of these parameters in ANSYS. The main focus of this paper is to analyze behavior of mechanism in different loading conditions and also results are verified by manual equations. A good design of mechanism must be effectively. The mechanism shall not work effectively but also be reliable in its strength and durability yet not over-design. In order to optimize analysis for exercise mechanism, first we need to simulate the Multibody dynamics analysis for load and stress predictions. Therefore In this paper, we introduce methodology to simulate and analysis the whole contact range of health exercise mechanism Multibody dynamics analysis by using ADAMS, ANSYS14 and CATIA V 5.

***Keywords**: CAD model, FEA, Vibrator Mechanism, MBD, ADAMS, CATIA, ANSYS.*

---***---

1. INTRODUCTION

The mechanism is used to convert rotary motion of electric motor to sliding motion of the mechanism. Due to that sliding motion of the mechanism whole body of man starts vibrating and any person doing exercise feel comfort. During the actual working different stresses are induced in links, main body that can be analyzed by using ANSYS 14 software and basically model has prepared by using CATIA V5 Software and also results are verified by using ADAMS software [ref.3] By ADAMS software we get different forces on different joints and different links. Also weight of the mechanism and different stresses on the mechanism we get by ANSYS 14.

2. MODELING

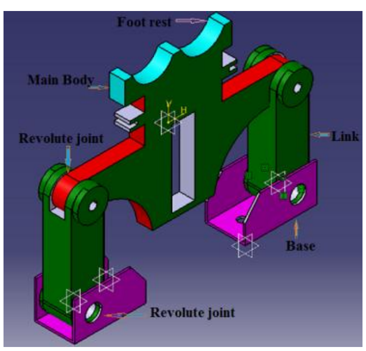

Fig 1. Catia V5 Assembly model

This model has prepared by using CATIA V5 software .It consist of main body having an adjustment for doing exercise and also external motion is applied to it.link is joined with main body by revolute joint and it has joined with base which is fixed. During the modeling in CATIA V5 we used different tools like part design, assembly design.

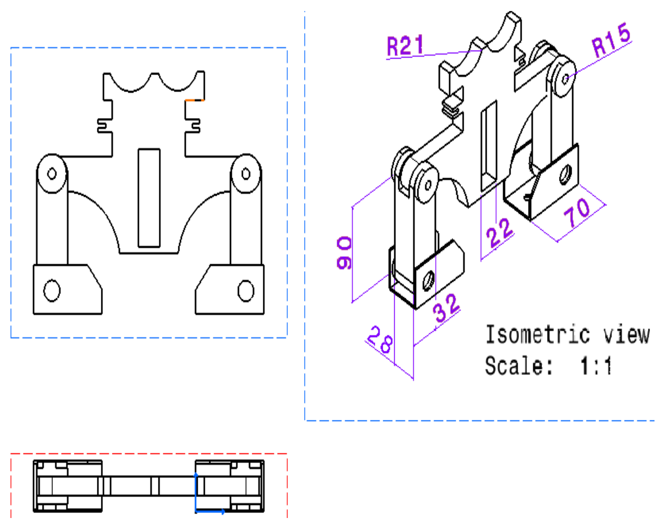

Fig 2 Catia V5 model with Dimensions

3. MATERIALS

The material used for this mechanism is Aluminum alloy. The results we get by ANSYS14 and also we are verifying by analytical equations.

Table-1: Constituent of Al2219 in Weight %

Elements	Mg	Si	Cu	Zr	Fe	Zn	Ti	V	Zn	Al
Weight in%	0.02Max	0.20 Max	5.8 - 6.8	0.1-0.25	0.30 Max	0.10 Max	0.02 -0.1	0.05 - 0.15	0.1 Max	Remaining

4. ANALYSIS BY ANSYS

In Ansys workbench we imported file from CATIA V5 in STP. Format for analysis of Mechanisms. Got different results by applying boundry conditions, material properties and meshing conditions for improved results. As shown in figures below.

Table 2 Geometry variables

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	G:\ASSEMBLYaaaaaaaaaaaaaaaaaa.stp
Type	Step
Length Unit	Meters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	0.23615 m
Length Y	3.9e-002 m
Length Z	0.18563 m
Properties	
Volume	4.1516e-004 m³
Mass	1.15 kg
Scale Factor Value	1.
Statistics	
Bodies	7

During the analysis this model has been imported from CATIA V5 stp format to ANSYS14.We applied material properties to this model that's Aluminum and also 25kg load is applied on main body which having support to do exercise.
We get different results like weight of mechanism, principal stresses, vonmisses stress, strains, and deformations .By this results we will easily understand the actual behavior of mechanism.

The messing model in ANSYS14 shown in following figure

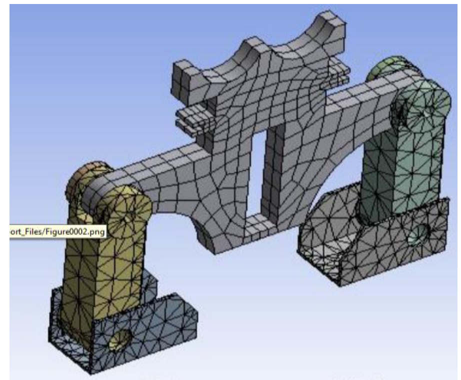

Fig 3. Meshed assembly in Ansys

Figure no.2 shows the meshing elements .In messing the whole mechanism is divided in to number of small parts. Due to that we get accurate results.

Table 3 Node & Elements for FEA

Statistics		
Nodes	3499	2385
Elements	1897	338
Mesh Metric	None	

Von misses stresses induced in this mechanism as shown in this figure below.

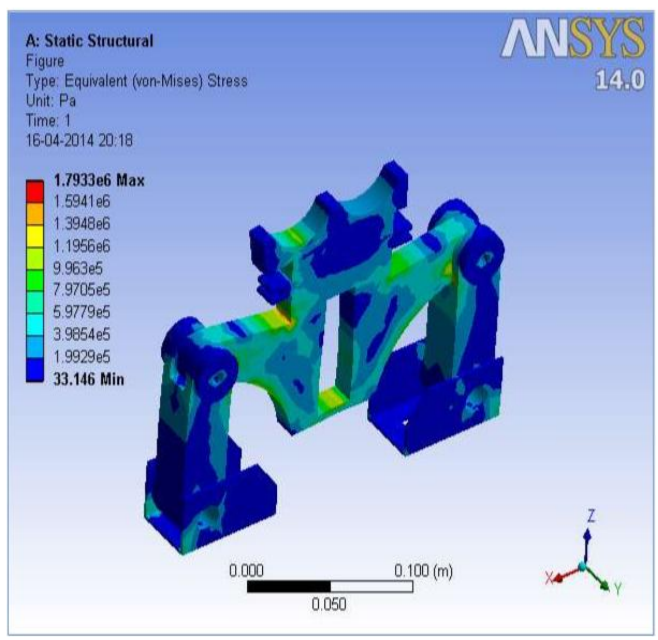

Fig.4 Von-misses stresses

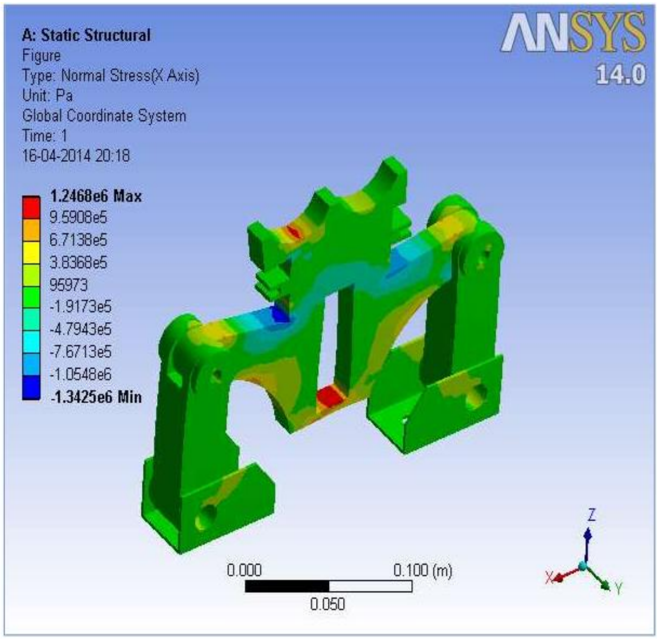

Fig.5 Normal elastic strain

Table 4 Real Constants for FEA

Properties		
Volume	9.1752e-005 m³	1.9768e-004 m³
Mass	0.25415 kg	0.54757 kg
Centroid X	5.151e-002 m	-4.7448e-002 m
Centroid Y	2.0169e-002 m	1.95e-002 m
Centroid Z	5.877e-002 m	0.1149 m
Moment of Inertia Ip1	2.6931e-004 kg·m²	1.8346e-003 kg·m²
Moment of Inertia Ip2	2.6937e-004 kg·m²	6.0134e-004 kg·m²
Moment of Inertia Ip3	4.2396e-005 kg·m²	1.2539e-003 kg·m²

5. ANALYSIS BY ADAMS

ADAMS stands for Automatic Dynamic Analysis of Mechanical Systems and was originally developed by Mechanical Dynamics Inc.(MDI). MDI was formed by researchers/developers of the original ADAMS code at University of Michigan, Ann Arbor, MI, USA. Later on, it was absorbed into McNeil Schindler Corp (MSC) in 2002.[ref.5]During the normal operation under 25kg of wait the different joints carries different loads during the normal operation with time .With the variation of time joints carries different loads. Has is shown in figure.

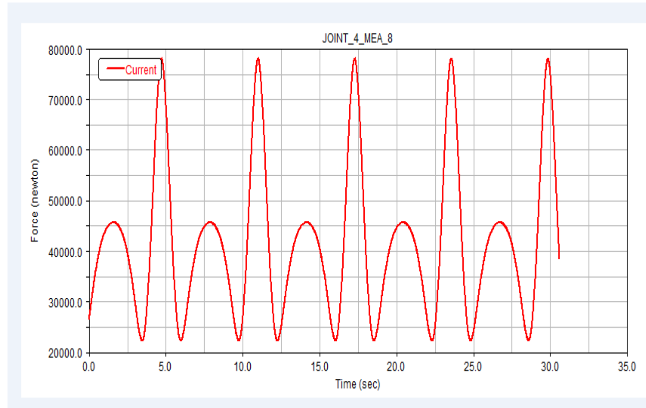

Fig.6. Forces acts on joints in different time in sec.

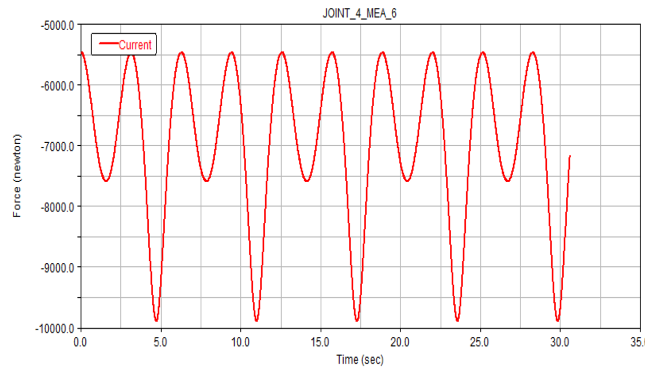

Fig.7 Forces acts on joints in different time in sec.

6. MATHEMATICAL CALCULATION

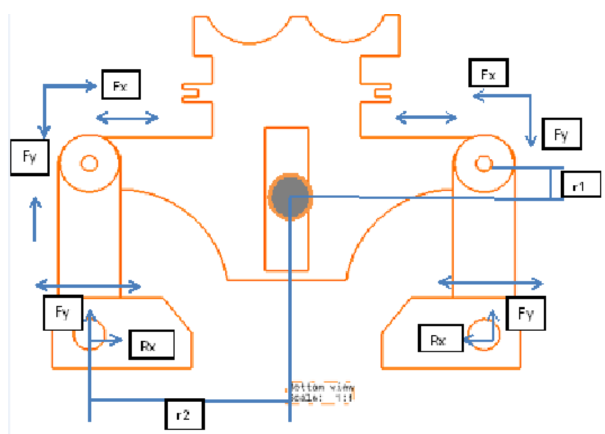

Fig.8 Forces acts on joints

Different variables

a=angular acceleration of link, a cm=Linear accretion of link=mass of the link, Fp=motor force, Cm=moment of inertia,T=Torque of the motor
Initially we are giving motion to left link

∑Fx=MAx
Fx-Rx=MAx
Fx=Rx+MAx--------------(1)

∑Fy=MAy
Fy-Ry=MAy
Fy=Ry+May-----------------(2)

T+Fxr1-Fyr2+Rxr1-Ryr2=Ia`---------------(3)

By solving these three equation we get matrix representation. In this equations Rx and Ry values we will get by external torque applied due to motor but Fx and Fy are the unknowns we get by matrix representation.

$$\begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ r1 & -r2 & 1 \end{bmatrix} \begin{bmatrix} Fx \\ Fy \\ T \end{bmatrix} = \begin{bmatrix} Rx+MAx \\ Ry+May \\ Ia-Rxr1+Ry \end{bmatrix}$$

Rx and Ry are the perpendicular distance we get .
Example

Fp is the external force due to motor
Fp acts 50N @ 90° (i.e.angle between horizontal surface and Link)

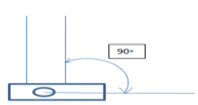

Fig.8 Forces at 90°

Then
Fx=cos0*50=50N
Fy=cos90*50=0N
On this basis we can calculate all forces are acting on machine frame

7. CONCLUSIONS

Analysis and simulation of health exercise vibrator by using Adams12 ANSYS14 and CATIAv5 was really very useful in analyzing multi body problem. In actual cases, most of the problems involve multi body system. Using these tool, engineers can evaluate virtual prototypes of complex physical problem and optimize designs for performance, safety and comfort, without the inevitable time-scale and cost risks in building and testing physical prototypes. Therefore, based on this paper, it is recommended to use above said softwares for designing and validating innovative products.

ACKNOWLEDGEMENTS

We express our first and fore most pranamas to his Holiness Dr.Channabasava Pattadevaru. President, SVES. The authors acknowledge with thanks to Principal Dr.Bipin Bihari Lal who made this Endeavour possible and also express our gratitude and indebtedness to Bheemanna Khandre Institute of Technology, BHALKI for providing us an opportunity to undergo research work successfully.

REFERENCES

[1]. Dr R. P. Sharma *; Chikesh ranjan *"Modeling and Simulation of Four-Bar Planar mechanisms Using Adams",IJMET, Volume 4, Issue 2, (2013), pp. 429-435
[2]. Retheesh Kumar,Akash Mohanty, " Design, Synthesis And Simulation Of Four Bar Mechanism For Eliminate The Plowing Depth Fluctuations In Tractors" IJERA Vol. 3, Issue 3, 2013, pp.953-956
[3]. Ryan, R.R., "ADAMS multibody system analysis software", MDI Technical Publications, Vol. 2, 1989.
[4]. TALABĂ, D., BATOG, I., "A method to Represent the Rigid Bodies Orientation for Multibody Systems Analysis", International Computer Science Conference microCAD '97", Miscolc, 1997, Section L, pp. 9-13.
[5]. VOLTERRA, E., ZACHMANOGLOU, E. C., "Dynamics of Vibrations", Charles E. Merrill Books, Inc., Columbus, Ohio, 1965