Design of a Flexural Joint using Finite Element Method

Abdullah Aamir Hayat, Adnan Akhlaq, M. Naushad Alam

Abstract

This paper presents the design and analysis of a compliant mechanism using hyperbolic flexural hinges. The flexural joint study is carried for the general Stewart platform with 6-6 configuration which has six degrees of freedom. The paper is aimed at designing a flexural joint using finite element method. First the maximum joint forces required for a particular movement of the Stewart platform is obtained from using MSC ADAMS software. The solid model developed in MSC ADAMS is validated with respect to the analytical Newton-Euler formulation for a set of motion. The results are in good match with each other for the extensions of legs. Then the forces are used to determine maximum stresses induced in the joints for different neck dimensions and lengths utilizing FEA software ABAQUS.

Keywords: Stewart Platform, Flexural Joint, Hyperbolic, FEM

1 Introduction

The purpose of this study is to consider the design and profile of the flexural hinge with high axial stiffness and high bending flexibility which can be implemented in the desired area with particular specifications. Flexure hinges are widely used in applications such as gyroscopes, accelerometers, balancescales, high accuracy alignment devices for optical fibre, high precision camera and multiplying linkages in Robotics. Furthermore, in the past decade micromanipulation has emerged as an important technological advancement and as a result the use of flexure hinges increased.

Various micro-motion stages were developed using conventional technologies based on servomotors, ball screws and rigid linkages. However, these conventional technologies encounter problems which struggle to achieve high positioning accuracy because of factors like FrictionWear, Backlash,low flexibility and high stress concentration and Lubrication etc. [1].On the other hand, compliant micro-motion stages, which solely move through deformations of flexure hinges, provide the capability of achieving highly precise positioning with kinematic stability stated by Handley et al. [2]. This is because of the fact that they are manufactured monolithically, they are very compact structurally. Flexural hinges with single-axis can be divided into two main categories: leaf and notch type hinges.

In 1965 Paros and Weisbord [1] introduced the first notch type hinge. Lobonitu et al. [3] introduced the parabolic and hyperbolic hinges configuration. Due to relative low rotation precision and stress concentration leaf type hinge is seldom adopted. Gui-Min et al. [4] represented the compliance model of the right circular hybrid flexural hinges.

The results in the paper will include the parametric study on optimum flexural design that will be performed by changing the length of flexure and the eccentricity

Abdullah Aamir Hayat
Mechanical Engineering Deptt., ZHCET AMU, Aligarh. Email: aamir_hayat@rediffmail.com

Adnan Akhlaq
Mechanical Engineering Deptt., ZHCT AMU,Aligarh. Email: adnanakhlaq87@gmail.com

M. NaushadAlam
Mechanical Engineering Deptt., ZHCET, AMU, Aligarh. Email: naushad7863@rediffmail.com

that defines the foci of hyperbola with a given boundary set in the structure that will be fixed in translation and rotation. Forces will be applied on the faces and the change in maximum stress pattern is observed. And the mode shape for the model is also presented.

2 Mechanism for Implementing Flexural Joint

The parametric model of the Stewart platform considered for the dynamic analysis for the forces at the joints of base and moving platform are: $[\boldsymbol{R_b} \quad \boldsymbol{r_m} \quad \boldsymbol{t} \quad \boldsymbol{\theta_b} \quad \boldsymbol{\theta_m}] = [0.60 \quad 0.45 \quad 0.25 \quad 0.3 \quad 0.5\,]$, where $R_b$is radius of base, r_m is radius of moving platform in meters, t is distance between platforms in meters and θ_b, θ_m are half joint base angle and moving platform angle in radian.

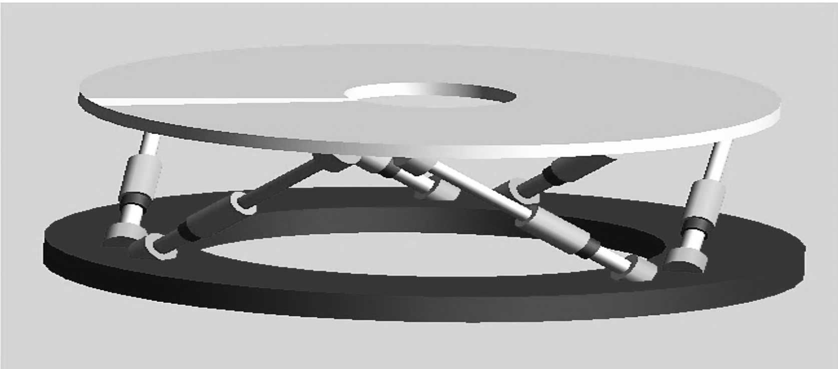

Figure 1: The Stewart platform manipulator built in MSC ADAMS. It has six prismatic actuators and fixed bottom plate and the moving upper plate is attached with the plunger with the spherical joints.

The position for the flexural joint in the above model is depicted in the Fig. (2) below. The radius of the plunger constrains the maximum thickness of the neck in flexural joint and the length is constrained by the geometry of the model.

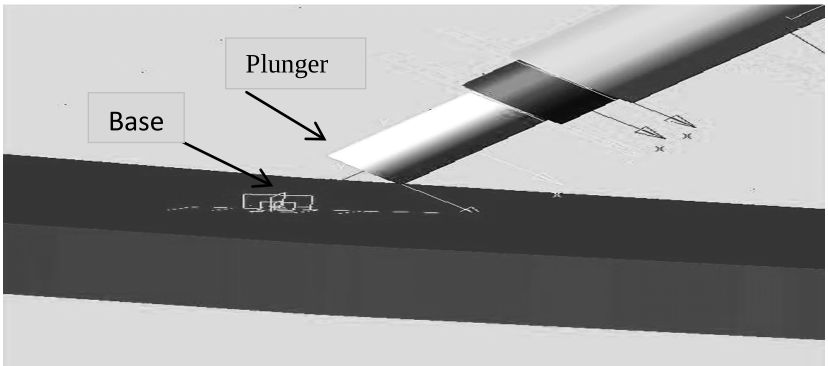

Figure 2: Position for the flexure joint at the bottom of plunger and fixed plate in the MSC ADAMS.

2.1 Dynamic analysis for the forces at joints

Based on the above kinematic parameter the model is developed in the MSC ADAMS and the validation is done by analytical formulation based on the Newton-Euler formulation. Validation problem considered is such that it would involve motion in all six dimensions. One such motion is described as: Let the center of mass of the moving platform is to follow a helical trajectory while rotating simultaneously about its CM in all three mutual orthogonal directions. The parametric description of such motion is stated below:

$$x = r_h \sin \omega_t\, t \;;\; y = r_h \cos \omega_t\, t \;\; ; z = p_h t$$

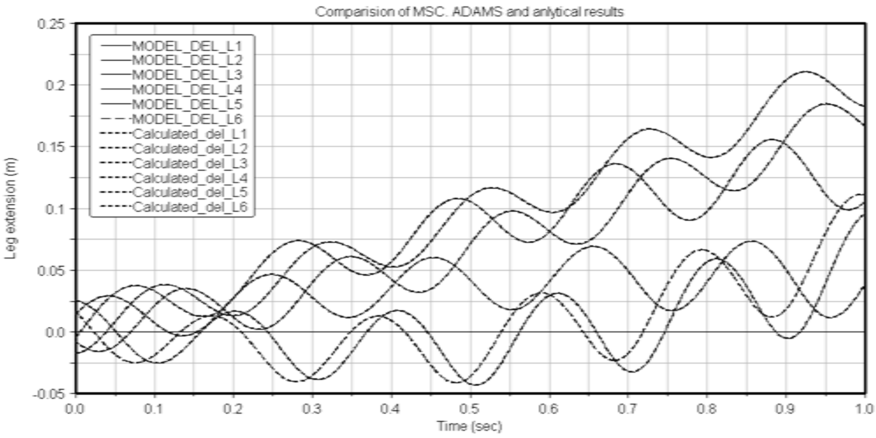

Figure 3: Comparision of Extensions of Legs for the motion stated above by MSC ADAMS and analytical results.

The elongation of the legs obtained through analytical solution and solid model, required to generate above parametric description of the helix is shown in the Fig. (3) The exact match of the results of two approaches validates the solid model.

2.1.1 Forces at the joints

Forces at the joints are calculated for the translation and orientation of the platform center of mass of the moving platform shown in Fig. (4). The maximum forces acquired at upper and lower spherical joints for the simulated trajectory presented in Table 1.

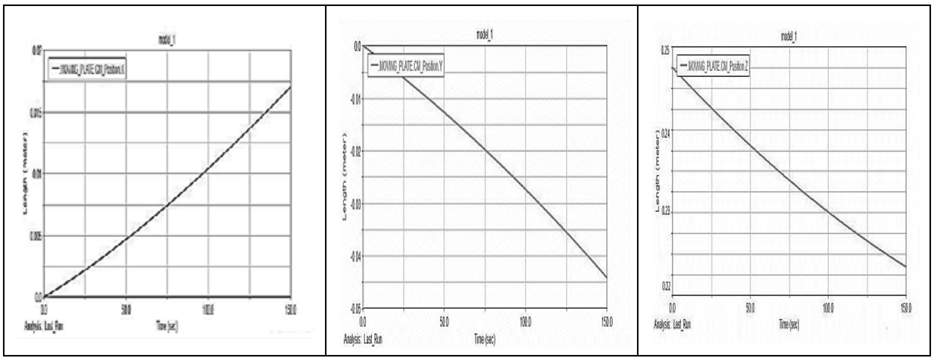

Figure 4:. Positions for CM of moving platform for x, y and z direction.

The ordinate in Fig. (4) are length in meters and on the abscissa there is the simulation time. The maximum amongthe magnitude of the forces in the x, y and z-direction is utilized for the design of the flexural joint.

Table 1Maximum forces acquired during simulation in the joints of six

Upperplatform Joint	**Lower platform Joint**
• X-axis 125Newton	• X-axis 250 Newton
• Y-axis 45 Newton	• Y-axis 290 Newton
• Z-axis 75Newton	• Z-axis 375 Newton

3 Design of Flexural Joint

For the analysis of flexure joint finite element modelling package ABAQUS was used. ABAQUS performs static and dynamic analysis and simulation on structures. It can deal with bodies with various loads, temperatures, contacts, impacts, and other environmental conditions.

3.1Design parameters

Titanium was selected as material for flexures since titanium and its alloy have high strength to weight ratio as well. Also the axial stiffness of the compliant mechanism relates to the working range. Because high axial stiffness results in a small working range, the compliant mechanism has to have low axial stiffness for a large working range.The property taken into account are listed below for the Titanium

Table 2: Propeerties of the material used for the Flexural joint

Modulus of Elasticity(N/mm^2)	Poission Ratio	Density (kg/m^3)	Yield strength(Pa)
109872	0.3	4500	1345E6

A standard form for the equation of a hyperbola with its centre at the origin is given as below

$$\frac{x^2}{a^2} - \frac{y^2}{b^2} = 1 \tag{1}$$

$$y = \sqrt{\frac{x^2 + t^2(e^2 - 1)}{(e^2 - 1)}} \tag{2}$$

From the above equations the hyperbola is plotted in MATHEMATICA which is shown in the Fig. 4. The eccentricity is taken as 25 for the hyperbola shown below. The directrix and the focii are also depicted.

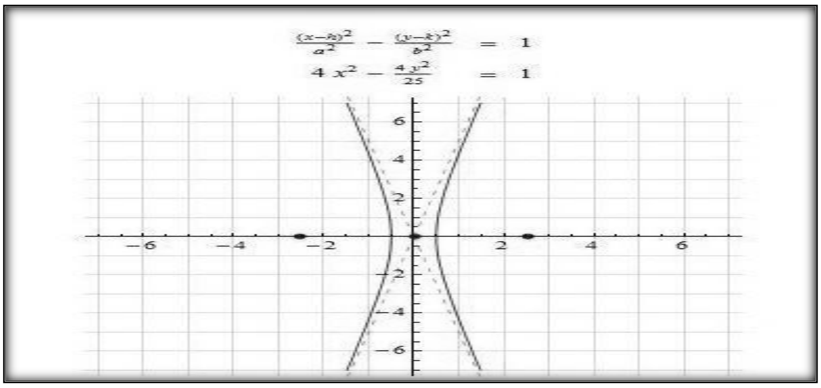

Figure 5: Hyperbola drawn in MATHEMATICA

The constraints in the design parameters for the flexural joint that can be used in the model developed with the specifications as T_1 _ _ $_2$ and $L_1 \le L \le L_2$.

where T and L are the thickness of the hyperbolic flexural joint at the neck and L is the length respectively.Here T_1 and T_2 are 0.4 to 20 mm respectively and L_1 and L_2 are 10 and 25 mm respectively.

4 FEM Analysis and Results

The meshed part with a tetragonal element C3D4 type with 59200 elements are taken into account as after the 50000 elements the variation in stress becomes negligible for this geometry. Then the boundary conditions are applied to the flexural hinge structure at the bottom as it is attached to the fixed base so the geometry at the bottom is fixed as shown in the Fig. (6).

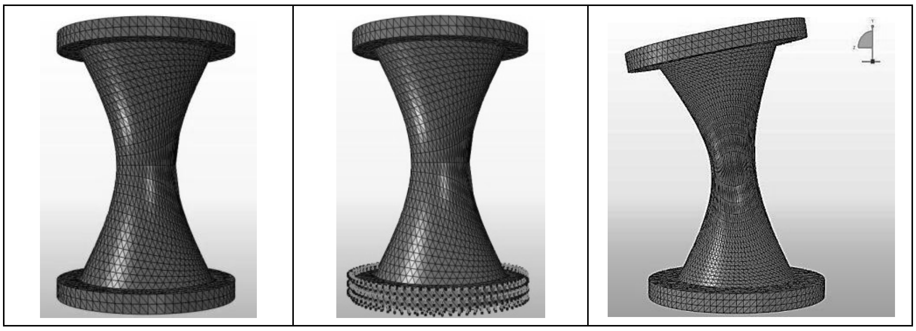

Figure 6: Meshed part, the boundary condition encaster at bottom and the bending of the flexure joint after application load.

The load is applied to the upper face which is obtained from the dynamic analysis and the bending of the structure for the given load was found within the moving angle of the platform and is 8.2 degree

The stress pattern obtained after the simulation is shown in Fig. (7). The stress pattern shows that the maximum stress occur at the mid region of the geometry and do not exceed the yield strength. Also there is a difference in stress at two ends of the mid-section due to the bending in the flexure.Now the parametric study was performed to have the value of maximum stress at different length and thickness for a constant Eccentricity.

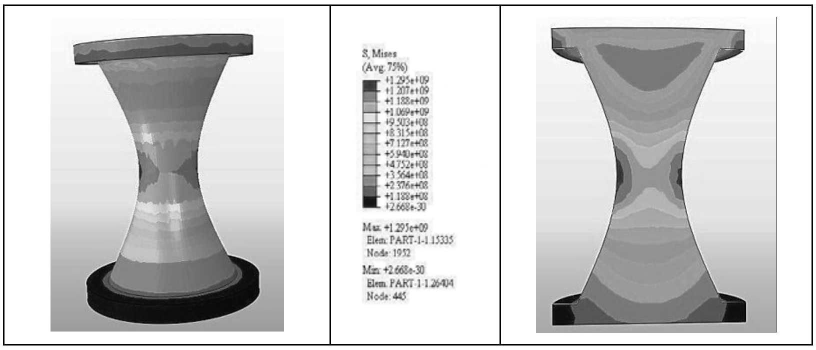

Figure 7: The stress contour for the applied load on the flexure joint

The variation of maximum stress with the change in length of the flexure is depicted below for the constant thickness of 6mm and the eccentricity as 25 is shown in Fig.(8).

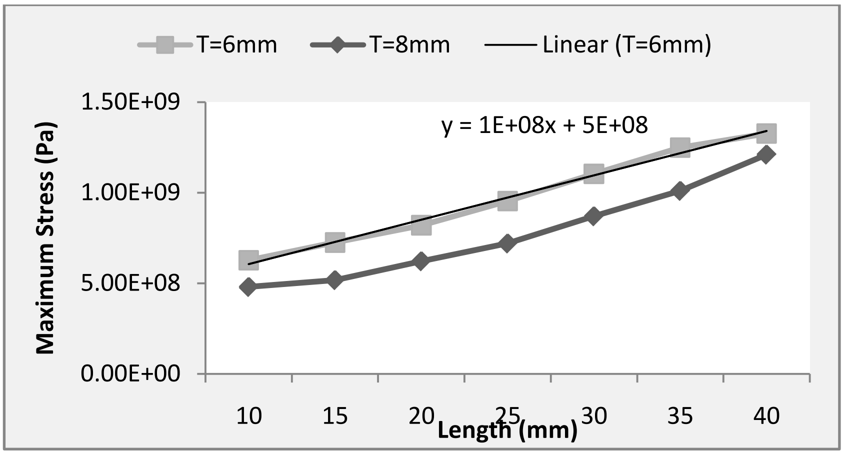

Figure 8: Stress dependence on Length and Thickness for constant eccentricity

Now varying the thickness of the flexure with constant length of 15mm the variation is depicted below. The linear fit variation for the stresses and the thickness is also depicted in the equation form in the Fig. (8).

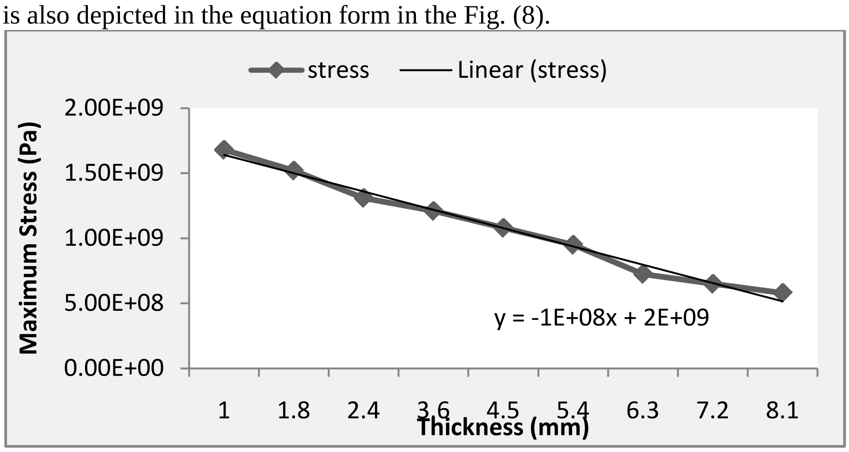

Figure 9: Stress dependence on Thickness for constant Length of the Flexural joint.

The first eight mode shapes of the flexural joint are depicted in the Fig. (10).

Figure 10: First eight mode shape of the Flexure Joint

5 Conclusions

The parametric analysis shows that as the thickness of the flexural joint is increased for a constant length the maximum stress which induces in the neck of hyperbola decreases, and for the increase in length for constant eccentricity and thickness the maximum stress induced also increases. The linear fit is also depicted in the graph for the data obtained after finite element analysis.

References

[1] J. Paros, L. Weisbord,"How to design Flexure Hinge,"*Mach. Des.*,vol. 37, pp.151–156, 1965.

[2] D. Handley, T.F. Lu, and Y. Yong,"Workspace investigation of a three DOF compliant micro-motion stage,"*Proceedings of the eighth international conference on control, automation, robotics and vision*, vol.84, pp.1279-1281 2004.

[3] N. Lobonitu, J.S.N. Paine, E. Gracia, and M. Goldfarb, "Corner-Filleted Flexural Hinges,"*ASME Trans. J. Mech. Des*., vol. 123, pp. 346-352, 2001.

[4] G. M. Chen, J. Y. Jia, and Z.W. Li, "On Hybrid Flexural Hinges", IEEE Transactions on Robotics, 2005

[5] K. Hari, and K.Srinivasan,*"Kinematic and Dynamic analysis of Stewart Platform-Based machine Tool Structures*", Robotica, vol. 21, pp.541–554, 2003.

[6] HKS. Abaqus explicit v6.4 user's manual. Abaqus Inc., 2003.