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

Modeling and analysis of Compliant Cantilever Beam

Akshay Vidap[1], Dr. Bhagyesh Deshmukh[2]

[1, 2] Solapur University, Walchand Institute of Technology, Ashok Chowk, Solapur 413006, India

Abstract: ***Micromechanisms are essential in the development of microsystems. Conventional mechanisms involve pin joints which have the typical limitations of backlash. To overcome issue of backlash a new class of mechanisms known as compliant mechanisms is developed. The design of compliant mechanism can be carried out using lumped design and or distributed design approach. In this paper, a detailed analysis and simulation of a cantilever beam (lumped System) is presented. Effect of radius variation and constraint condition on the lumped cantilever beam is also carried out.***

Keywords: Compliant mechanism lumped and distributed approach, Cantilever beam.

1. Introduction

Recent development in microsystems increases dement of micromechanisms. In the microfactory precise and controlled motion is required for micromanipulation and microassembly applications. The conventional mechanisms are uses rigid body mechanisms with pin joints in motion transmission application. These mechanisms are least suitable for the microapplication because it has some limitations such as pin joints, backlash, wear and tear etc. to overcome these issues a different class of mechanism is developed known as compliant mechanism. In the compliant mechanism structural properties are used for obtaining desired output. Due to various advantages such as no backlash, less wear and tear, least joints or jointless features it may be provide the solution for the precise and controlled motion applications. Due to its no of advantages over the conventional mechanisms it has vast area of application. They are used in the various fields such as compliant motion amplifier in Micro Electro Mechanical Systems (MEMS) [1-4], robotics [5] etc. Wind shied wiper used in the automobile is made up of assembling no of parts but with the help of compliant mechanism it is possible to integrate all system in one part (monolithic construction).

2. Design Approaches

The concept of the pseudo-rigid-body modeling technique and other approaches has successfully opened the way for simple design and analysis of many compliant mechanisms. Two approaches are widely used by researchers for design compliant mechanism.

a) **Kinematic synthesis approach**

It is based on traditional rigid- body kinematics called as pseudo-rigid body mechanism. The purpose of the pseudo-rigid body model (PRBM) is to provide a simple method of analyzing system that undergoes large nonlinear deflections [2]. Design is carried out by synthesizing a rigid body mechanism. Then design is made flexible with the help of flexible hinges and checked it for its predetermined work. The pseudo-rigid body model predicts the motion as well as force- deflection relationship. Pin joints and springs are used in the model. The important part is finding out places of pin joint and spring constant. The PRBM has been used almost exclusively for design and modeling of products where the elastic deflections are in the nonlinear range, but dynamic effects are not a large factor in mechanism performance. The PRBM has a proven track record as a reliable predictor of static behavior for compliant mechanisms. The power of the PRBM is its ability to convert a difficult to analyse compliant mechanism into a familiar rigid-body mechanism which can be analysed using traditional kinematic approaches. The nature of the PRBM will not allow the static model to capture the higher Eigen modes for high driving frequencies. However, the ability to model compliant mechanisms in frequency ranges of interest using kinematic models (from the PRBM), provides valuable insight into the behaviour of the devices that is not available from finite element models. This insight is critical for analysis and making engineering decisions for improving device performance [6].For Small length flexural pivots the spring constant k is given by [2]

Small length flexural pivots the spring constant k is given by [2]

$$K = \frac{(EI)l}{l}$$

Where,

l = Length of flexible segment

E= Young's modulus

I = Beam moment of inertia

Flexure Hinge as shown in Figure 1, was treated as a torsional spring and taking stiffness was taken as [7],

$$K_\theta = \frac{2Ebt^{2.5}}{9\pi r^{0.5}}$$

Where,

K_θ= Torsional stiffness of the hinge

E = Young's modulus of material used

b = Thickness of plate used

t = Hinge thickness

r = Flexure radius

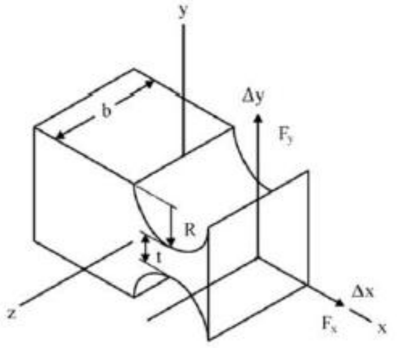

Figure 1: Flexure hinge [7]

In the pseudo rigid body modeling deflection is given by

$$Y = \frac{2Fl^2}{k_\theta}$$

By using above formula displacement is calculated and tabulated below

b) Continuum synthesis approach

It is based on topology optimization. In this approach whole mechanism is considered as a continua and analysis and synthesis is carried out. It allows the change of topology and movement of mechanism is obtained via the distributed elastic deformation of the whole mechanism [2]. There are two methods used in topology optimization, one is Ground structure parameterization and other is Continuous material density parameterization. The first step in the design of a compliant mechanism is to establish a kinematically functional design that generates the desired output motion when subjected to prescribed input forces. This is called topological synthesis. Although the size and shape of individual elements can be optimized to a certain extent in this stage, local constraints such as stress and buckling constraints cannot be imposed while the topology is being determined. Once a feasible topology is established, performance constraints can be imposed during the following stage in which size and shape optimization are performed. Performance constraints may include minimizing the energy loss in the mechanism, obtaining desired motion amplification (geometric advantage) or force amplification (mechanical advantage), or ensuring that none of the elements buckle under the action of applied forces and external loads [8].

3. Analysis by Numerical Method

After PRB Modeling of the cantilever beam and solving it by analytical method which was based on replacing the hinge by rotational spring and using the equation of torsional stiffness, the same models are then compared by using a numerical method software ANSYS

3.1 Finite Element Model

In this study only planar compliant mechanisms are considered and it has been ensured that the Compliant Mechanisms are made from material sufficiently thick to support the x-directional load such that out of plane displacements are insignificant. Therefore, out of plane displacements can be ignored so that only translations x-y plane need be considered. FEA models have been commonly used to model compliant mechanisms. The 2-D Finite Element Model (FEM) is well suited to the Cantilever beam.

3.2 Steps in FEM

Following are the generic steps undertaken in the Finite Element Analysis (shown in Figure 2) for a compliant Cantilever beam. Each of these steps corresponds to specific processor/processors within the Processor Level in ANSYS. Model generation is done in the Pre-processor and then after the application of loads and the solution of problem is performed in the Solution Processor. The results are viewed in the General Postprocessor.

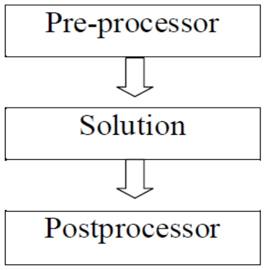

Figure 2: General steps in ANSYS

There are several other processors within the ANSYS program. The above mentioned steps are undertaken in the analysis of Cantilever beam and are described in this section.

The basic modeling of the mechanism was performed using the Pseudo Rigid Body Model (PBRM) concept and classical cantilever beam theory to simplify the model and to predict the mechanism response under prescribed input motion.

a) ANSYS Pre-processor:

Ansys pre-processor comprises of the generic steps mentioned below

i. Create the model geometry
ii. Define materials/material properties
iii. Generate finite element model (mesh)

i. Create the model geometry

To begin with the analysis process, the Model generation is carried out in this processor.
A solid Model is generated in ANSYS as shown in Figure 3.

Figure 3: Cantilever beam model

ii. Define material properties

This step enables to define the material to be used and associated mechanical properties. In the present work, structural steel is used as material for design of a compliant cantilever beam. The Young's modulus and Poisson's ratio for structural steel is taken as 2X105 N/mm2, 0.3 respectively.

iii. Generate the mesh

As the material properties for the material selected are defined for the solid model of the compliant cantilever beam, the mesh generation is carried out and it is shown in Figure 4. Flexure meshed using fine mesh.

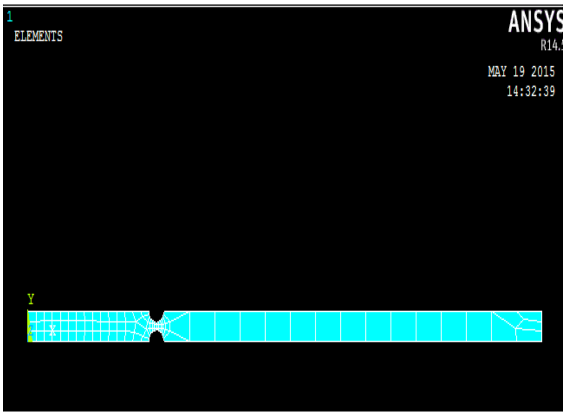

Figure 4: Meshed model

b) ANSYS Solution Processor

The meshed model is imposed with the boundary conditions and the load steps for the detailed displacement (deformation) and stress analysis.

i. Define analysis type and analysis options

In the current work, one of the prime objectives is to attain the micro displacements. As the scope of the study and tune with the objectives the main focus is on proper displacement with reducing stress concentration. The 'static structural analysis' module of ANSYS is considered for the analysis. The static analysis focused on aspects such as force and displacement relationships.

ii. Specify boundary conditions

After selection of the analysis type, it is then required to impose the boundary conditions for the analysis. Figure 5. Show the application of the boundary conditions to the model.

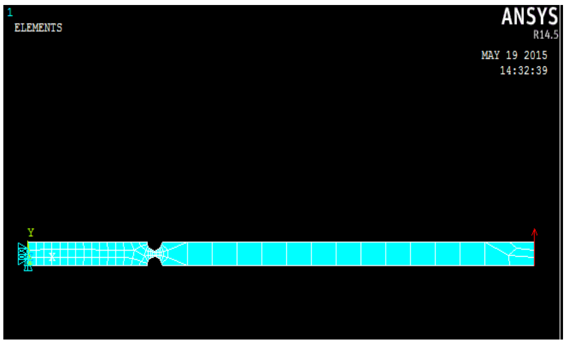

Figure 5: Fixed support

iii Obtain solution

This step initiates the solution process of model of flexure based microstage. The solution is carried out for resulting deformation and induced stress. The compiler processes the model and keeps the solution ready for the further processing.

c) ANSYS General Postprocessor

The solution obtained by the solution process is a general solution. The Displacement plot is obtained for the particular load condition and is represented in Figure 6

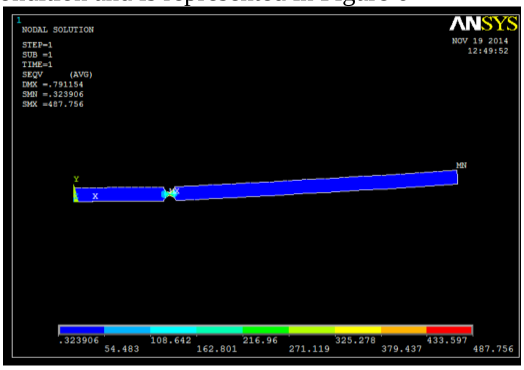

Figure 6: Deflection of lumped beam.

A convergence study has also been carried out. Procedure is repeated for length variation and the results for analytical and simulations are illustrated in Table 1.

Table 1: Flexure length vs Displacement

Force (N)	*Flexure Length from fixed link (mm)*	*Deflection (mm)*		*Stress (N/mm2) by simulation*
		Analytical	*Simulation*	
1	75	0.927556	0.801619	506.836
1	50	0.412249	0.521933	337.157
1	25	0.103062	0.354029	168.883

4. Analysis of Effect of Radius Variation of Flexure Profile

A joint-less cantilever beam with defined force displacement relation is taken as a design problem for the study. The flexure parameters are taken in a range from r =1.5-0.1-0.6 mm, which results in neck thickness t=1-0.2-2.8 mm. The analysis is carried out using following approach.

For actuation force F is kept constant as 1 N. Simulation of lumped beam is carried out by FEA analysis for flexure radius of 1.4 mm. Deflection result of the model is shown in Figure 7 for 1 N force and maximum deflection is observed to be 0.599666mm.

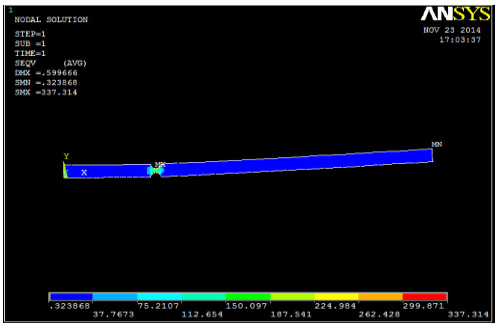

Figure 7: Deflection of lumped beam.

Similar procedure is repeated for flexure radius range of 1.5-0.1-0.6 mm and neck thickness range as 1-0.2-2.8 mm and the results by analytical method and by simulation are obtained and illustrated in Table 2.

Table 2 : Flexure radius vs Deflection for lumped system with upper and lower end unconstraint

Force (N)	Flexure Radius (mm)	Deflection (mm)		Stress(N/mm2) by simulation
		Analytical	Simulation	
1	1.5	0.9274	0.8016	487.756
1	1.4	0.5679	0.5996	337.31
1	1.3	0.3722	0.4923	247.07
1	1.2	0.2561	0.4270	188.48
1	1.1	0.1827	0.3882	176.74
1	1.0	0.1338	0.36	148.01
1	0.9	0.1000	0.3417	119.161
1	0.8	0.0759	0.3282	100.347

From the Table 2 it is observed that below the flexure radius 1.3mm there is variation in deflection results and also sudden change in stress results. This is because as the flexure radius varies below 1.3 mm, instead of deformation in the flexure region it is observed that the complete beam deforms. It is a case of distributed system.

5. Result and discussion

Compliant mechanism can be designed by two approaches namely kinematic synthesis approach and Continuum synthesis approach. PRB modeling is used mostly for the lumped system analysis. Displacement of the lumped cantilever beam in depends on the flexure displacement. Variation of flexural length is carried out and it is observed that as length of flexure increases from the fixed end there is reduction in displacement. Effect of radius variation on the displacement of the cantilever beam is carried out. It is observed that below the flexure radius 1.3 mm there is variation in deflection results and also sudden change in stress results. This is because as the flexure radius varies below 1.3 mm, instead of deformation in the flexure region it is observed that the complete beam deforms. It is a case of distributed system.

6. Acknowledgement

Authors are thankful to the management, Principal Dr. S. A. Halkude and Prof. A. B. Ankulkar of Walchand Institute of Technology, Solapur for their continuous encouragement and support.

References

[1] P. khare, G.B, Madhab, C.S. Kumar, and P. K. Mishra. ‘‘Optimizing design of piezoelectric actuated compliant microgripper mechanism’’, 13th National Conf. Mech. Mach., Bangalore, India (2007), pp1-4.
[2] L. L. Howell “Compliant Mechanisms” Mechanical engineering department Brigham Young University, John Wiley & Sons, Inc. (2001) pp. 1-18.
[3] S.Solehuddin , M.I.Z. Ridzwan and A H. Kadarman “Methodology of Compliant Mechanisms and its Current Developments in Applications: A Review” School of Mechanical Engineering, University Sains Malaysia, 14300 Nibong Tebal Seberang Perai, Penang, Malaysia American Journal of Applied Sciences 4 (3) (2007), pp.160-167.
[4] S .k Nah, and Z. W. Zhong, ‘‘A microgripper using piezoelectric actuation for micro-object manipulation’’, J. Sens. Actuators, A133, (2007), pp. 218–224.
[5] B. Trease (2004) “Flexures” A Lecture Summary, Compliant System Design Laboratory, University of Michigan.
[6] Midha, A., Howell, L.L., Norton, T.W., 2000. Limit positions of compliant mechanisms using the pseudo-rigid-body model concept. Department of Mechanical and Aerospace Engineering and Engineering Mechanics, University of Missouri Rolla, Rolla, MO, 65409-0050, USA, Mechanism and Machine Theory 35, 99-115.
[7] Yong, Y.K., Tien-Fu Lu, Handley, D.C., 2008. Review of circular flexure hinge design equations and derivation of empirical formulations. School of Mechanical Engineering, The University of Adelaide, SA 5005, Australia, Precision Engineering 32, 63–70.
[8] Kota, S., Rordgers, S.M. and Sniegowski, J., 2001. Design of Compliant Mechanisms: Applications to MEMS. Analog Integrated Circuits and Signal Processing, 29, 7–15.

Author Profile

Akshay Vidap completed B.E.(Mechanical) pursuing M.E.(Mechanical Engineering Design) from Walchand Institute of Technology, Solapur, Solapur University.