Finite Element Analysis of Kinematic Linkage

R.R.Kolhe[1] Prof.S.S.Pimpale[2]
[1]PG Student [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]Rajarshi Shahu College of Engineering, Pune University

Abstract— This research paper presents modification in the four bar linkage that will offer a variable output for constant input in addition to the advantages of high efficiency. Thus objective of project is defined to develop a variable displacement mechanism that will convert constant rotating motion into oscillating motion that can be varied for the same constant input by using manual control. Accordingly the parts are designed and some standard pars are designed. Then 3D CAD model is created in Pro-Engineering to define the functional dimension analysis and Fit function analysis and FEA is performed in ANSYS workbench. FEA is performed to make sure that the component of the linkage is safe for critical loading and the prototype can be manufactured. Here in this paper presents the basic of finite element method and example of linkage which is solved in the ANSYS.

Key words: FEA, Variable Displacement Linkage, FEA of Kinematic Linkage

I. INTRODUCTION

A. General Procedure of FEA

In this finite element analysis the continuum is divided into a finite numbers of elements, having finite dimensions and reducing the continuum having infinite degrees of freedom to finite degrees of unknowns. It is assumed that the elements are connected only at the nodal points. The accuracy of solution increases with the number of elements taken. However, more number of elements will result in increased computer cost. Hence optimum number of divisions should be taken. In the element method the problem is formulated in two stages:

1) The Element Formulation

It involves the derivation of the element stiffness matrix which yields a relationship between nodal point forces and nodal point displacements.

2) The System Formulation

It is the formulation of the stiffness and loads of the entire structure. Certain steps in formulating finite element analysis of a problem are common to all such analyses whether structural, heat transfer, fluid flow or some other problem. These steps are embodied in commercial finite element software packages.

B. Basic steps in the finite element method

1) Discretisation of the domain

The geometry (solid model) is divided into a number of finite elements by imaginary lines or surfaces. The Interconnected elements may have different sizes and shapes .The success of this idealization lies in how closely this discredited continuum represents the actual continuum. The choice of the simple elements or higher order elements, straight or curved, its shape, refinement is to be decided before the mathematical formulation starts.

2) Identification of variables

The elements are assumed to be connected at their intersecting points referred to as nodal points. At each node, unknown displacements are to be prescribed. They are dependent on the problem at hand. The problem may be identified in such a way that in addition to the displacement which occurs at the nodes depending on the physical nature of the problem, certain other quantities such as strain may need to be specified as nodal unknowns for the element, which however, may not have a corresponding physical quantity in the generalized forces. The value of these quantities can however be obtained from variation principles.

3) Choice of Approximating Functions

After the variables and local coordinates have been chosen, the next step is the choice of displacement function, which is the starting point of mathematical analysis. The function represents the variation of the displacement within the element. The function can be approximated in many ways. A convenient way of expressing it is by polynomial expressions.

The shape of the element or the geometry may also approximate. The coordinates of corner nodes define the element shape accurately if the element is actually made of straight lines or planes. The weightage to be given to the geometry and displacements also needs to be decided for a particular problem.

4) Formation of Element Stiffness Matrix

After the continuum is discretised with desired element shapes, the element stiffness matrix is formulated. Basically it is a minimization procedure. The element stiffness matrix for majority of elements is not available in explicit form. They require numerical integration for this evaluation. The geometry of the element is defined in reference to the global frame.

5) Formation of the Overall Stiffness Matrix

After the element stiffness matrix in global coordinates is formed, they are assembled to form the overall stiffness matrix. This is done through the nodes which are common to adjacent elements. At the nodes the continuity of the displacement functions and their derivatives are established. The overall stiffness matrix is symmetric and banded.

6) Incorporation of Boundary Conditions

The boundary restraint conditions are to be imposed in the stiffness matrix. There are various techniques available to satisfy the boundary conditions.

7) Formation of the Element Loading Matrix

The loading inside an element is transferred at the nodal points and consistent element loading matrix is formed.

8) Formation of the overall loading matrix

The element loading matrix is combined to form the overall loading matrix. This matrix has one column per loading case and it is either a column vector or a rectangular matrix depending on the no. of loading conditions.

9) Solution of simultaneous equations

All the equations required for the solution of the problem is now developed. In the displacement method, the unknowns are the nodal displacement. The Gauss elimination and Choleky's factorization are most commonly used methods.

10) Calculation of stresses or stress resultants

The nodal displacement values are utilized for calculation of stresses. This may be done for all elements of the continuum or may be limited only to some predetermined elements.

II. VALIDATION

Although validation is not a formal part of the FEM analysis, its inclusion in the process is very important. Blindly trusting an FEM simulation without first checking its correctness can be dangerous. The validation step usually involves comparing FEM results at one or more selected positions with exact or approximate solutions. These exact or approximate solutions are derived using classical approaches covered in mechanics of materials or elasticity courses. Carrying out the validation step also strengthens conceptual understanding and enhances learning.

In order to become a skillful FEA user, a thorough understanding of techniques for modeling a structure, the boundary conditions and, the limitations of the procedure, are very crucial. Engineering structures, e.g., bridge, aircraft wing, high-rise buildings, etc., are examples of complex structures that are extremely difficult to analyze by classical theory. But FEA technique facilitates an easier and a more accurate analysis. In this technique the structure is divided into very small but finite size elements (hence the name finite element analysis). Individual behavior of these elements is known and, based on this knowledge; behavior of the entire structure is determined.

FEA solution of engineering problems, such as finding deflections and stresses in a structure, requires three steps:

- Pre-process or modeling the structure
- Analysis
- Post processing

A brief description of each of these steps follows.

A. Step 1: Pre-process or modeling the structure

Using a CAD program that either comes with the FEA software or provided by another software vendor, the structure is modeled. The final FEA model consists of several elements that collectively represent the entire structure. The elements not only represent segments of the structure, they also simulate it's mechanical behavior and properties. Regions where geometry is complex (curves, notches, holes, etc.) require increased number of elements to accurately represent the shape; whereas, the regions with simple geometry can be represented by coarser mesh (or fewer elements). The selection of proper elements requires prior experience with FEA, knowledge of structure's behavior, available elements in the software and their characteristics, etc. The elements are joined at the nodes, or common points. In the pre-processor phase, along with the geometry of the structure, the constraints, loads and mechanical properties of the structure are defined. Thus, in pre-processing, the entire structure is completely defined by the geometric model. The structure represented by nodes and elements is called "mesh".

B. Step 2: Analysis

In this step, the geometry, constraints, mechanical properties and loads are applied to generate matrix equations for each element, which are then assembled to generate a global matrix equation of the structure. The form of the individual equations, as well as the structural equation is always,

$$\{F\} = [K]\{u\}$$

Where

$\{F\}$ = External force matrix.

$[K]$ = Global stiffness matrix

$\{u\}$ = Displacement matrix

The equation is then solved for deflections. Using the deflection values, strain, stress, and reactions are calculated. All the results are stored and can be used to create graphic plots and charts in the post analysis.

C. Step 3: Post processing

This is the last step in a finite element analysis. Results obtained in step 2 are usually in the form of raw data and difficult to interpret. In post analysis, a CAD program is utilized to manipulate the data for generating deflected shape of the structure, creating stress plots, animation, etc. A graphical representation of the results is very useful in understanding behavior of the structure.

The above described software procedure is mostly transparent to the user. A user has the following interaction with the software, through user's computer.

- Create the geometry, representing the structure: A CAD modeling software is used to create the structure's geometry.
- Provide the material properties, loads, constraints, etc.
- Analyze the result data.

D. Concept of Eccentric Pivot Mechanism Using Four Bar Kinematic Linkage

This mechanism is to convert rotary motion of crank into oscillatory output of the output element. The angle of oscillation of the output element depends on position of pivot element. The pivot element position can be varied because it is placed on a slide. Thus adjustment of the piston stroke can be done by varying the position of the pivot element. This mechanism is selected because it offers maximum stability and vibration-less performance and nature of mounting of the pivot element which is mounted on a screw slide with accurate adjustment of pivot position permits continuously variable position of pivot and therefore stroke of the output. The mechanism is basically an inversion of four bar kinematic linkage.

III. METHODOLOGY

A. CAD Model

Schematic illustration of variable displacement linkage Assembly is shown in figure

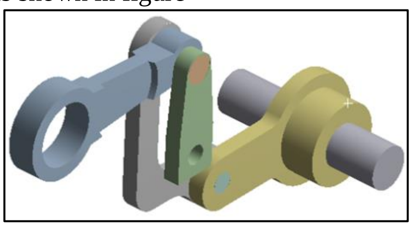

Fig. 1: CAD model imported in ANSYS Workbench

B. Finite Element Model

The Finite Element Method only makes calculations at a limited (Finite) number of points and then interpolates the results for the entire domain (surface or volume) by using shape function. Any continuous object has infinite degrees of freedom and it is not possible to solve the problem in this format. The Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements).The model was meshed with element solid 185. SOLID185 is used for the three-dimensional modelling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, stress stiffening, large deflection, and large strain capabilities. The FE model of variable displacement linkage shown in figure 2.

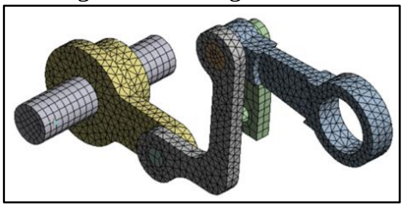

Fig. 2: FE model of linkage

C. Boundary Condition

To do analysis, it necessary to give the boundary conditions. The displacement constraint of shaft, is fixed in all direction which is shown in the figure 3. The force constraint applied on the connecting rod is shown in figure 3.

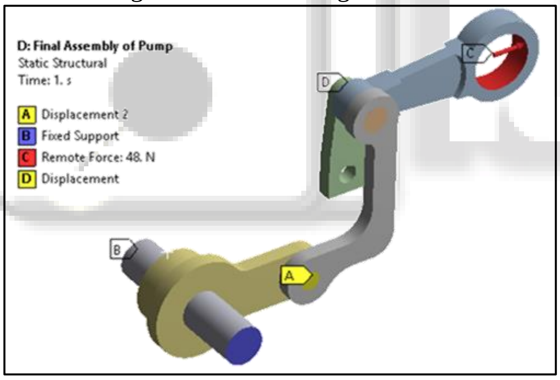

Fig. 3: Boundary conditions

D. Solution

In this stage the matrices are generated and solved for the unknown variables. This part is fully automatic. The FE solver can be logically divided into three main parts; the pre-solver, the mathematical engine and the post-solver. The pre-solver reads the model created by the pre processor and formulates the mathematical representation of the problem. All parameters defined in the pre processing stage are used to do this so if something is left out the pre solver will complain and cancel the call to the mathematical engine if the model is correct the solver proceeds to form the element stiffness matrix for the problem and calls the mathematical engine which calculates the result (displacement, temp and pressures etc.). The result are returned to the solver and the post solver is used to calculate strain, stresses, heat fluxes, velocity etc) for each node within the component or continuum. All these result are sent to a file which may be read by the post processor.

E. Step 3: Post Processing

Post-processing is the most important step in analysis. You may be required to make design decision based on the result. Post processor is used to review the result carefully and check the validity of the solution.

1) Deformation

It gives quick indication whether load were applied in correct direction or not. Figure 4 gives total deformation plots.

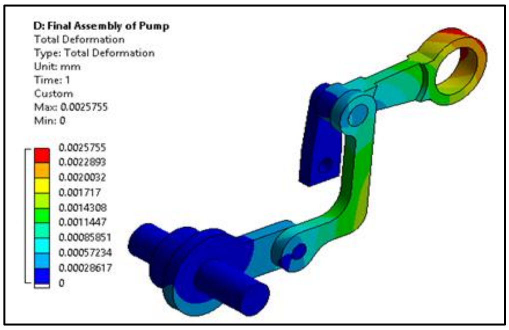

Fig. 4: Total Deformation

2) Equivalent Stress plots

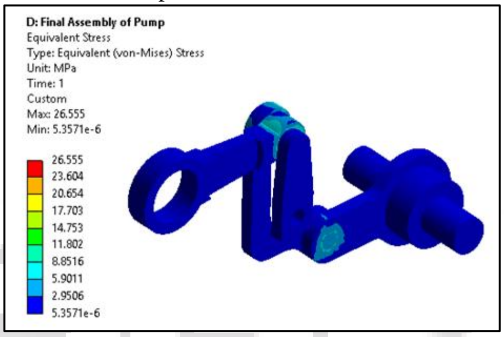

Fig. 5: Stress Distribution

3) Equivalent Strain

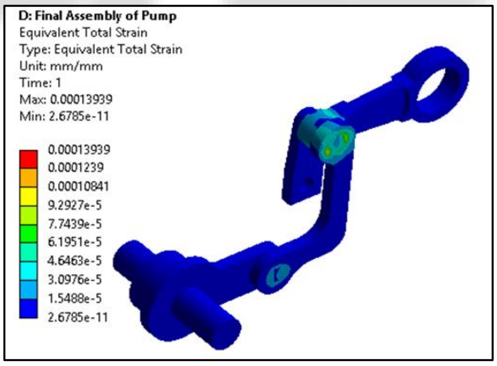

Fig. 6: Strain Distribution

4) Frictional or Contact Stress

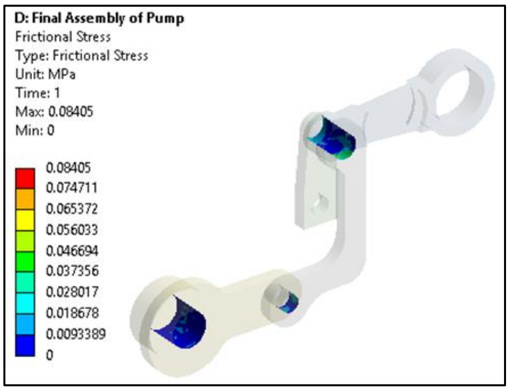

Fig. 7: Frictional stress

IV. CONCLUSION

The main objective of the present development is to provide a variable displacement linkage which offers variable output from constant rotational input.

The design and development of variable displacement linkage is successfully developed and evaluated through FEA. Proposed design is safe because the stresses induced are below the yield strength and can be used where variable output is desired from constant input.

ACKNOWLEDGMENT

I wish to express my sincere thank to Dr. A.A. Pawar, HOD of Mechanical Department. for guidance and encouragement in carrying out project work. I express my sincere thanks to MR. N. P. Warke and MR. S.D. Sutar.

REFERENCES

[1] Dimitrios Xenos , Peter Grassl (2016), “Finite Elements in Analysis and Design” Vol.(117),pp.11-20
[2] Bowden F P and Tabor D (2001), “Friction and lubrication of Solids”, Vols. 1, 2,Clarendon, Oxford.
[3] Mahdavian S M, Mai Y W and Cottrell B (1982), “Friction, Metallic Transfer and Debris Analysis of Sliding Surfaces”,Wear, Vol. 82, pp. 221-232.
[4] Shawn R. Wilhelm and V James D. Van de Ven. “Synthesis Of A Variable Displacement Linkage For A Hydraulic Transformer” Proceedings of the ASME 2011 International Design Engineering Technical Conferences &Computers and Information in Engineering Conference IDETC/CIE 2011, Washington, USA, August 28-31, 2011.
[5] Georges E.M. Vael, Peter A.J. Achten and Titus van den Brink Innas BV. “Efficiency of a Variable Displacement Open Circuit Floating Cup Pump”, The 11th Scandinavian International Conference on Fluid Power, Linkoping, Sweden, June 2-4, 2009.
[6] Noah D. Manring Yihong Zhang. (2001,SEPTEMBER) The Improved Volumetric-Efficiency of an Axial-Piston Pump Utilizing a Trapped-Volume Design.volume (213), pp. 479-487.
[7] V. B. Bhandari, “Design of machine element”, Third edition ,Tata Mcgraw Hill Education,2010
[8] PSG Design data book compiled by PSG College of technology, Coimbatore.
[9] Ferdinand freudenstein; E. Roland maki, “variable displacement piston engine”, US Patent 4 270 495, Jun.2,1981
[10] Bradley D. Goodell, “variable displacement pump system”, US Patent 4 518 320, May21,1985
[11] H. Allen Myers; Kerry Geringer, “Manual displacement control”, US Patent 5 122 037, Jun.16,1992

