FINITE ELEMENT ANALYSIS OF STRESSES IN A TOOTHED WHEEL

BY

S. A. UGWU

MECHANICAL ENGINEERING DEPARTMENT

UNIVERSITY OF NIGERIA,

NSUKKA

ABSTRACT

Toothed wheels generally and gears in particular are in constant use in machine operation. Gears under operation, such as in automobile, are subjected to stresses of varying magnitude and dimension.

These stresses set up in a spur gear under operation are analyzed in this paper using Finite Element Method, FEM. The gear models were produced and loaded in a specially built gear loading system to determine the load being transmitted by the wheels.

The dimensions of the wheel were taken and plotted on graph sheet for the purpose of discritization. After the sub-division of the gear tooth into 242 triangular elements (constant strain triangle-CST), each element data namely the elastic properties, co-ordinates and connectivity of each element and the associated load components, were taken down. Care was taken to refine the mesh at the expected regions of stress concentration.

Then a mainframe computer cyber 932-11 was used to run the programme written and the results displayed.

1.0 Introduction

Numerical methods consist of the Boundary Element Method, the Finite Difference Method and the Finite Element Method FEM. In this work Finite Element Method FEM is applied. Numerical Methods for the solution of different equations have become particularly popular in recent times because modern technical problems require complex loading land shapes which seldom allow a closed solution.

It is believed that numerical application in elasticity started as early as 1906, but was not efficiently applied until the introduction of computer hardware and software. From 50s, numerical methods are firmly established in Engineering. The numerical methods are now so efficiently used that they have rendered some analytical and some experimental procedures obsolete.

Finite Element Method, in particular, offers a unifying approach to the solution of diverse engineering problems. Two basic features make the FEM to be superior to other competing methods. First, a geometrically complex domain of the problem is represented as a collection of geometrically Simple subdomains called finite elements. Second, over each finite element the approximations are derived using the basic idea that any continuous function can be represented by a linear combination of algebraic polynomials.

2.0 Finite Element Method

The basis of FEM is the element stiffness matrix. It involves dividing physical systems such as structure, solid, e.t.c, into a finite number of relatively simple elements whose load-displacement characteristics are known, can easily be derived or approximated using well tried method in the theory of elasticity. These elements are connected at a discrete number of points along their periphery known as nodal points.

When a problem is being solved with the use of FEM, it is important to be aware of the physics and engineering assumptions of it. This method offers a unifying approach to the solution of divers engineering problems. It is Superior to the other numerical methods as mentioned in the introduction.

2.1 The Element Stiffness Matrix:

As said above, any Structure being analyzed with the use of FEM may be broken down into finite number of relatively Simple elements whose load-displacement characteristics are known, can easily be derived or approximated. Having done this, the structure or component is then re-assembled using the cho-

sen elements and the overall or structure stiffness equations thus derived. These equations linking the structure loads and structure displacements are approximate; the degree of approximation depending on how well the assembly elements accommodate the loading and boundary conditions and obey the rules of compatibility. Hence, the element stiffness matrix is the basic building block of the FEM.

Let us take for instance, an arbitrarily shaped element with 'n' nodes as shown in fig. 1.

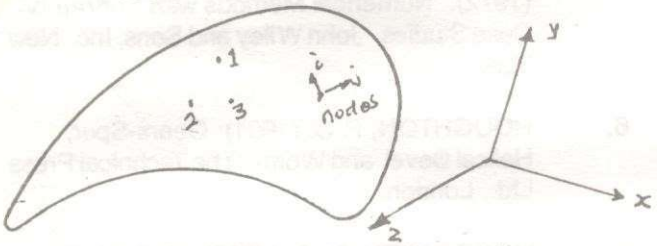


Fig.1: Nodal points on Arbitrarily Shaped Element. Each node is, in general, free to move in several directions and thus each node will have a number of element displacement (δ) associated with it and with each displacement will be associated an element load (P). Each element load P_i will depend upon every element displacement ($\delta_1, \delta_2, \ldots, \delta_n$) and for a linear elastic solid a typical equation would be:

$$P_1 = K_{11}\delta_1 + K_{12}\delta_2 + \cdots + K_{1n}\delta_n \quad \cdots (1)$$

where i = 1, 2, ... , n. The Coefficients K_{ij} in equation (1) are called Stiffness Coefficient and the whole of n equations would be written

$$\left.\begin{aligned} P_1 &= K_{11}\delta_1 + K_{12}\delta_2 + \cdots + K_{1n}\delta_n \\ P_2 &= K_{21}\delta_1 + K_{22}\delta_2 + \cdots + K_{2n}\delta_n \\ &\vdots \\ P_n &= K_{n1}\delta_1 + K_{n2}\delta_2 + \cdots + K_{nn}\delta_n \end{aligned}\right\} \ldots (2)$$

Equation (2) written in short matrix form gives:

$$\{P\} = [K]\{\delta\} \quad \ldots\ldots\ldots\ldots\ldots (3)$$

where { P } is the column vector of displacements and [K] is the element stiffness matrix.

Calculation of Element Stiffness matrix [K] is the starting point of FEM.

3.0 Calculation of Stresses with the Use of FEM:

In calculating stresses using an in-plane constant strain triangle CST Finite Element Method, FEM Computer programme (ref. 3) was employed. A mainframe Computer Cyber 932-11 was used to run the computation.

The programme which was completely listed is divided into three main parts, namely:-

(i) Calculation of the element stiffness matrices and assemblage into the over-all structural stiffness matrix.

(ii) Solution of the Set of simultaneous linear equation to determine the nodal point displacements.

(iii) Evaluation of stresses in the individual elements.

The flow chart of the programme (fig. 2.) is enclosed.

After the subdivision of the structure (figure 3) into 242 triangular elements (CST) as shown in figure (3), each element data, namely the elastic properties, the coordinates, the connectivity of each element and the associated load components were completely listed.

Care was taken to refine the mesh at expected regions of stress concentration, that is at the root of the tooth and the point of load application.

Regularly, as in FEM procedure, the tooth was now fixed by means of several supports (at nodes 48, 1, 4, 6, 8, 11, and 66). The load (282.71N) was applied at a pressure angle of 25° at nodal point 122. It is well known that two values of pressure angle are used for spur gears. According to Standards, these angles are 14.5° and 20° , the latter being the choice in most modern constructions. This pressure angle 20° , however, could increase due to tooth correction, centre distance shifting, etc. (refs. 4 and 6). It was noticed during the time of loading the gear on the gear loading system that the working pressure angle was 25° and such working pressure angle was used in the analysis. The technique applied, however, is suitable for any standard or working pressure angle.

The face width (thickness of gear blank) of the tooth (5mm) compared to the thickness of the tooth measured along the pitch line (20mm) and its length (26mm) justifies the assumption of the plane state of stress and therefore the use of an in-plane FEM analysis.

The accuracy of FEM Calculation depends on the number of elements used. It was checked that discretization into 242 elements is sufficient for the analysis having used Coarser meshes.

The analysis was carried out in a single pass using the input data which provided a geometrical description of the structure and its loading. This produces an output consisting of tabulated nodal point deflections and elements' stresses (not shown due to bulky nature of the print-out).

On the print-out stress components $\sigma_x, \sigma_y, \tau_{xy}$ principal stresses σ_1 and σ_2 and even $\sigma_1 - \sigma_2$ were shown.

The nature of CST elements implies that these stresses are constant throughout each Single triangular element (normally represented at the centriod of the element).

Three areas of maximum stress concentration could be noticed from the print-out. They are at point of Load application.

σ_2 = -52.59MPa (element 212) at the point of load application.

σ_1 = +23.37 Mpa (element 104) at the left fillet.

σ_2 = -20.24 Mpa (element 89) at the right fillet.

4.0 Conclusion and Recommendation

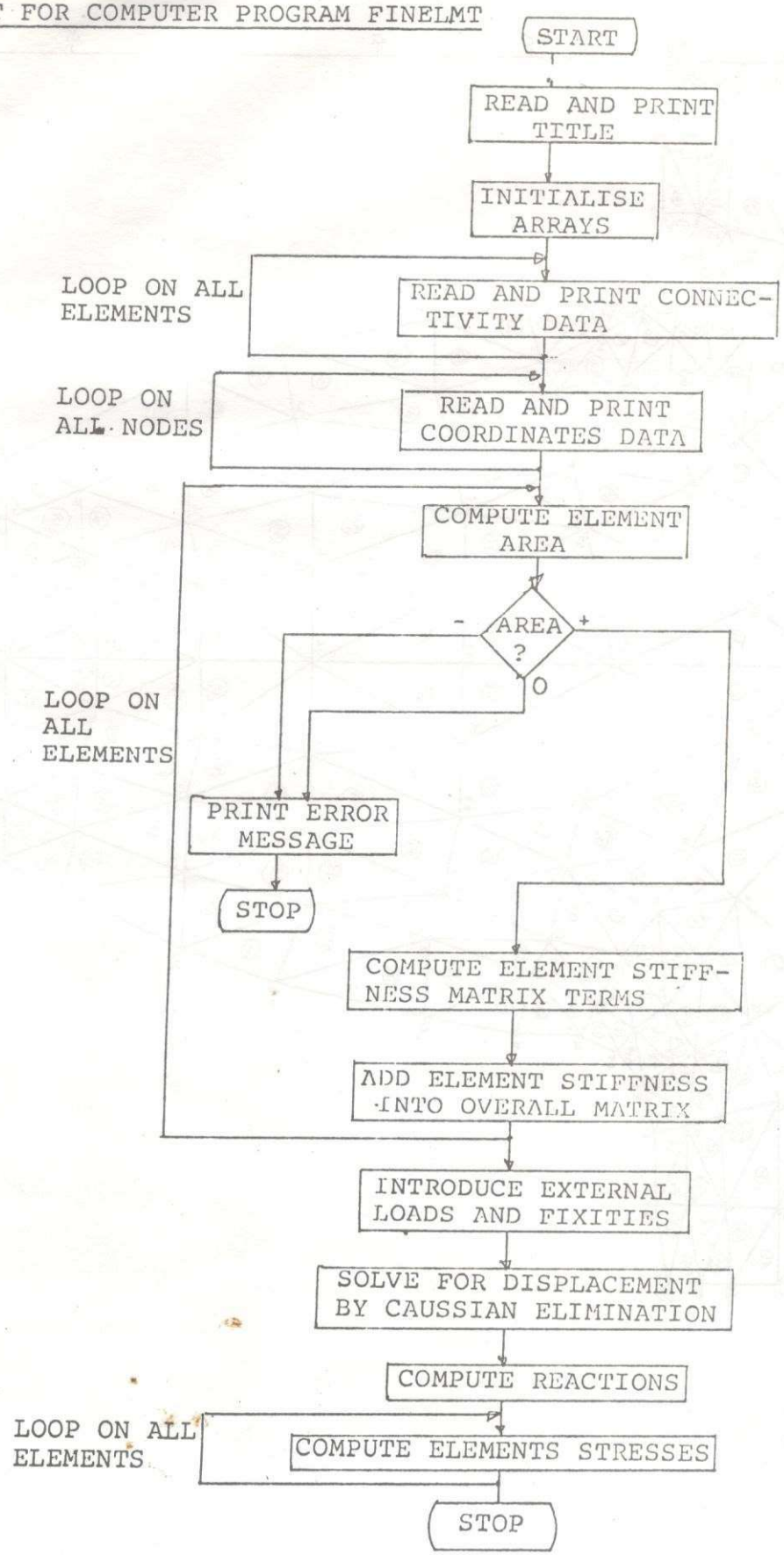
The Stresses in a toothed-wheel (spur gears) have been successfully determined using Finite Element Method, FEM. The results show that the in-plane, plane stress analysis is suitable for spur gears. In case of very thick gears, the FEM in-plane analysis could easily be replaced by a 3-D analysis. The advantage of using FEM is that a complete stress calculation (stress separation) is possible as could be seen from the print-out. Other well-known classical design formulae such as Lewis, Sopwith equations, etc, could be used to find stresses at the fillets and point of contact but not at any desired point on the tooth. Supposing a hole is drilled on the tooth for the purpose of weight reduction, FEM technique can be used to find the effect of the hole on the strength of the tooth.

This work shows that when a pair of gears are meshing, high stress concentration is noticed at the fillets and point of contact, being highest at the contact point. Shear stresses at these regions are relatively high too. Because of this stress concentration at the fillets, it is recommended that the gear designers should make the fillet radius as large as possible to avoid tooth breakage due to high stress concentration. The best results are obtained with the largest radius, that is, the semi-circular fillet.

References:

1. BORESI, A. P., BOTTOM. O. M. et al (1978): Advanced Mechanics of Materials (3rd Ed). John Wiley and Sons, New York.
2. *CHAPMAN, W. A. J. (1981); Workshop* Technology (part 3). The English Language Book Society and Edward Arnold (publishers) Ltd., London.
3. CHEUNG, Y. K. and YEO, M. F. (1979): A Practical Introduction to Finite Element Analysis. Pitman Publishers Ltd., London.
4. DONALD, J. MYATT (1962): Machine Design- An Introductory Text: MCGRAW Hill Book Company Inc., New York.
5. DORN, W. S. and MCCRACKEN, D. D. (1972): Numerical Methods with Fortran IV Case Studies. John Wiley and Sons, Inc. New York.
6. HOUGHTON, P. S. (1901): Gears-Spur, Helical Bevel and Worm. The Technical Press Ltd., London.
7. MCCRACKEN, D. D. (1972): A Guide to Fortran-IV Programming. John Wiley and Sons, New York.
8. MITCHEL, D. and LANCASTER, P. R. (1980): Advanced Solid Mechanics - Theory, Worked Examples and Problems. The Macmillan press Ltd., London.
9. PAUL, H. BLACK (1955): Machine Design. McGraw-Hill Book Company, Inc. New York.
10. ROAK (1954): Formulas for Stress and Strain. McGraw-Hill Book Company, New York.
11. ZIENKIEWICZ, O. C. and HOLISTER, G. S (1965): Stress Analysis - Recent Development in Numerical and Experimental Methods. John Wiley and Sons Ltd., London.

FLOW CHART FOR COMPUTER PROGRAM FINELMT



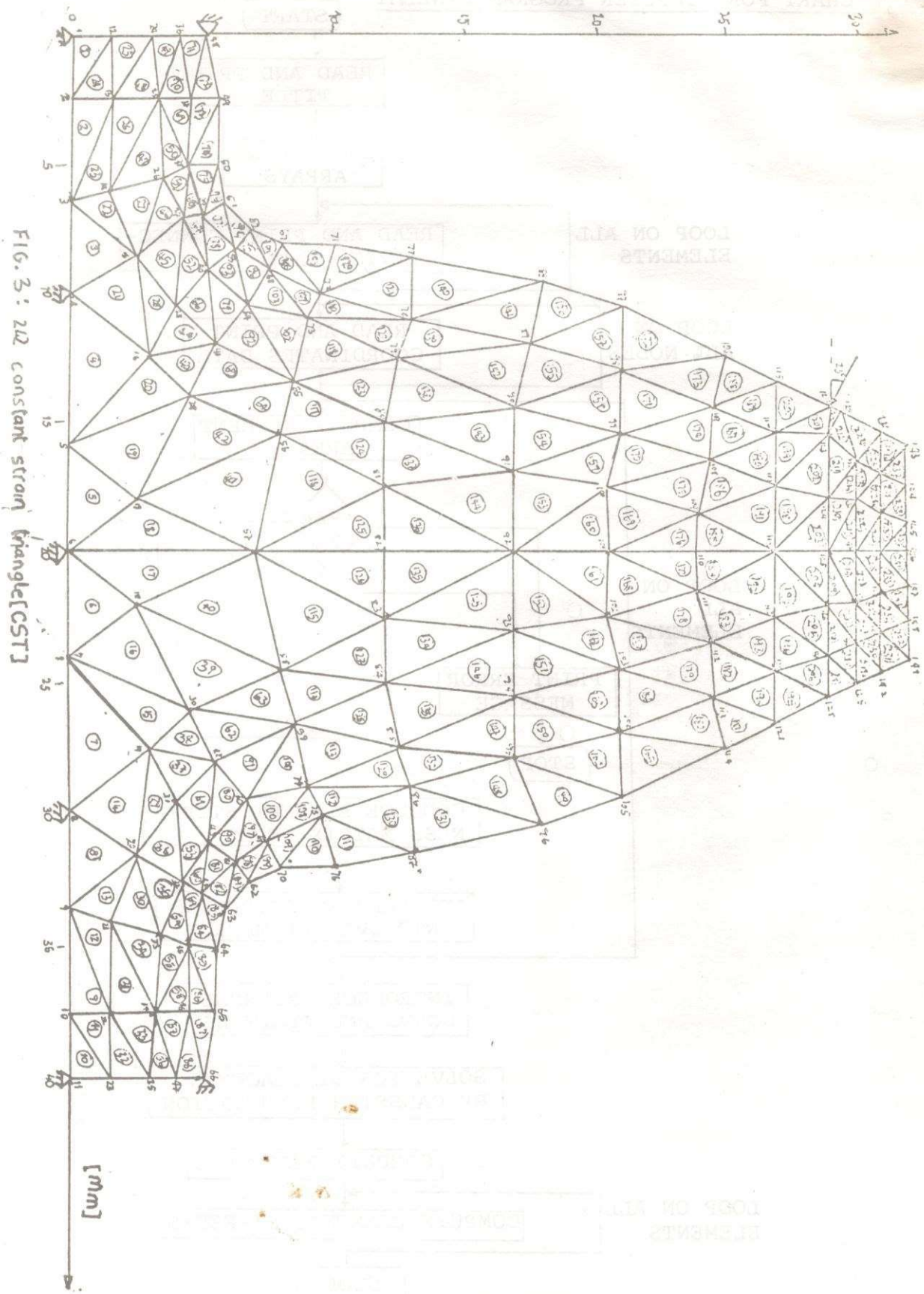


FIG. 3: 2D constant strain triangle (CST)