RESEARCH ARTICLE OPEN ACCESS

Finite Element Based Stress Analysis of Seat Belt Using Integrated Force Method

Anjali Kale[1], Dr. G. S. Doiphode[2]

[1]M.E. Student (Department of Applied Mechanics, The Maharaja Sayaji rao University of Baroda, Vadodara Gujarat-390001, INDIA)

[2]Associate Professor (Department of Applied Mechanics, The Maharaja Sayaji rao University of Baroda, Vadodara Gujarat-390001, INDIA)

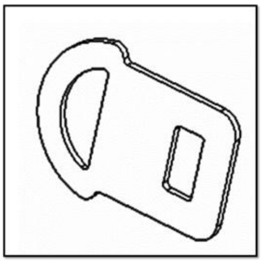
ABSTRACT

Integrated force method (IFM) has been developed for solving various continuum mechanics problems, where all the internal forces are taken as independent variables. The system equilibrium equations (EE's) are mathematically concatenated with the MATLAB based compatibility conditions (CC's) to form the global set of equations. This paper explains the solution strategy of prototype seat-belt component using integrated force method. Actual seat belt component is taken here to verify IFM based FE element are readily used which is named as REC_5F_8D and REC_13F_16D where 5F and 13F reflects number of internal unknowns and 8D and 16D are number of total displacements respectively. The results for "Von Mises Stress" and Maximum Displacement using IFM based formulation and compared with ANSYS.

Keywords – IFM, Seatbelt Component, REC_5F_8D, REC_13F_16D, ANSYS

I. INTRODUCTION

An Integrated Force Method (IFM) was proposed by Patnaik (1973) for the analysis of discrete and continuous systems. As, the present stiffness method dominates the structural analysis field. IFM provides alternative formulation to the stiffness method [4]. In this paper the seat belt tongue part is analyzed by 2D plane stress analysis. Proposed structures are as shown in Fig.1 a and Fig. 1 b.



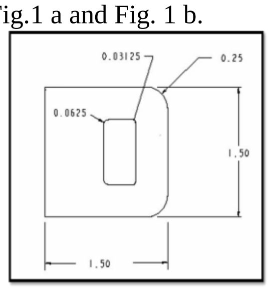


Fig.1 a seat belt component Fig.1 b seat belt tongue (all dimension in inch)

IFM based readymade elements REC_5F_8D and REC_13F_16D are used for this type of two-dimensional problem of structural mechanics using Integrated Force Method.

In Past IFM have been successfully applied for cantilever deep beam problem using REC_5F_8D element [1] using IFM in two dimensional category. Also, Circular plate with different boundary conditions using triangular elements has been analyzed using continuum analysis by IFM [3]. Determination of **"Von Mises Stress"** is also worked for both elements methods.

II. BASIC THEORY OF INTEGRATED FORCE METHOD

The main behavioral parameters of either discrete or continuum structures are the internal forces, and displacements. For the purpose of analysis the model can be designated by two attributes **'n'** and **'m'**, such as structure **(n, m)**. The number of force unknowns or force degree of freedom (fdof) is **'n'**. Likewise; the number of displacement degree of freedom (ddof) is **'m'**. Once 'n' forces {F} are evaluated, 'm' displacements {X} can be back calculated from the known forces and vice versa. From the theory of elasticity, Saint Venant formulated the compatibility conditions in the field of an elastic continuum, but he overlooked the conditions on the boundary, [5] which have since been formulated by S. N. Patnaik. Augmentation of Saint Venant's conditions with these boundary compatibility conditions completed the classical formulation, which is referred as "Completed Beltrami Michell's Formulation (CMBF)".

The basic concept of IFM is expressed by following symbolic equation:

$$\left[\frac{[B]}{[C][G]}\right]\{F\} = \left[\frac{P}{\delta R}\right] \text{ Or } [S]\{F\} = \{P\}^{*} \quad (1)$$

Where, [B] is the (m × n) equilibrium matrix,
[C] is the (r × n) compatibility matrix,
[G] is the (n × n) concatenate flexibility matrix,
{P} is the m-component load vector, and
{δR} is the r-component effective initial deformation vector,

From forces {F} the displacements can be back calculated using the following formula;

$$\{X\} = [J]\{[G]\{F\} + \{\beta^0\}\} \qquad (2)$$

; Where [J] = m rows of $[[S]^{-1}]^T$.

Thus, main important task in IFM based analysis is to develop the following three major governing matrices;

a) The equilibrium matrix [B], which works as a link between internal forces and external loads;
b) The compatibility matrix [C], which governs the deformations by [C]{β} = 0, and
c) The global flexibility matrix [G], which relates deformations to forces.

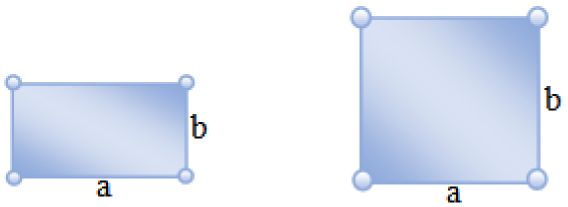
STEPS FOR THE ANALYSIS:

Step 1– Generating Equilibrium Equations
Step 2– Deformation Displacement Relations
Step 3– Compatibility Conditions
Step 4– Force Deformation Relations
Step 5– Coupling EE and CC

III. DEVELOPMENT OF [B], [C] AND [G]

Equilibrium Matrix [Be]:

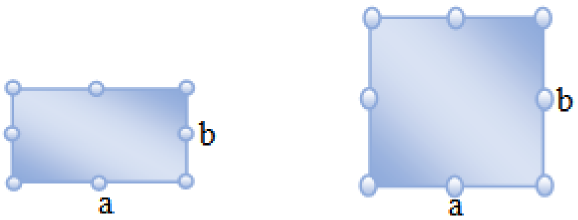
Using MATLAB as programming tool for each element, considering coarser discretization of tongue part of seat belt by using FE based REC_5F_8D and REC_13F_16D respectively as shown in following Fig.1 c and Fig.1 d. [2]



(1) (2)

b = 0.1718 inches **a = 0.1875 inches**
b = 0.2031 inches

Fig.1 c Two different sizes of REC_5F_8D



(1) (2)

a = 0.1875 inches **a = 0.1875 inches**
b = 0.1718 inches **b = 0.2031 inches**

Fig.1 d Two different sizes of REC_13F_16D

Compatibility Matrix [CC]:

The compatibility matrix is obtained from the deformation displacement relations ($\{\beta\} = [B]^T\{\delta\}$) for REC_5F_8D element. In the DDRs, 35 deformations (which correspond to the 35 force variables) are expressed in terms of 22 displacements ($\delta 1$, $\delta 2$,…., $\delta 22$). It has 13 CCs that are obtained by eliminating the 22 displacements from the 35 DDRs. In REC_13F_16D element contains 91 deformations (which correspond to the 91 force variables) are expressed in terms of 59 displacements. It has 32 CCs that are obtained by eliminating the 59 displacements from the 91 DDRs. All the matrix operations are carried out on the command prompt. By writing proper path for the .m file it is connected to the folder "CCPROG" on the desktop. Operating the module of "mtechexamplemod(B)" demands number of the compatibility conditions for making the global equilibrium matrix square as depicted in following Fig.1 e and Fig.1 f.

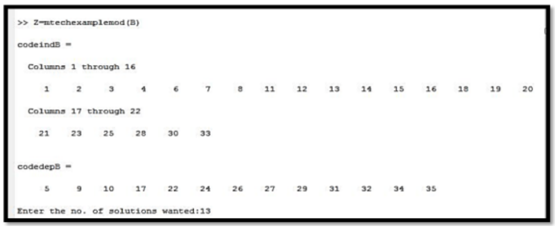


Fig.1 e operation related to compatibility conditions for REC_5F_8D

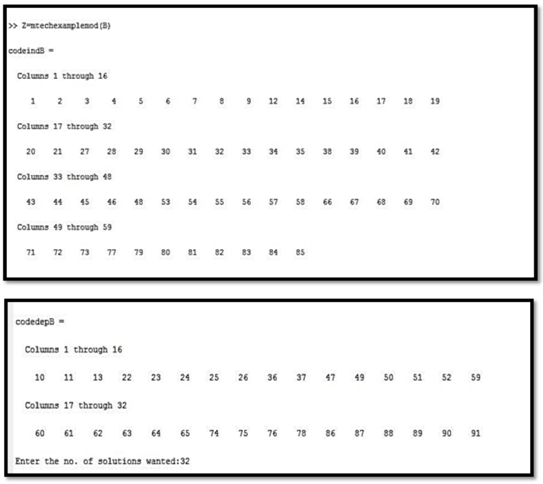


Fig.1 f operation related to compatibility conditions for REC_13F_16D

Flexibility Matrix [Ge]:

Using **MATLAB** as programming tool generation of flexibility matrix (Ge) for each element, considering coarser discretization in tongue part of seat belt two sizes were taken into account. The size of the flexibility matrix is of (5 × 5), which is depends on modulus of elasticity, thickness of the element and material properties respective.

IV. ANALYSIS OF SEAT BELT

The basic seat belt component after applying horizontal symmetry is shown in Fig.1 g and Fig.1 h.

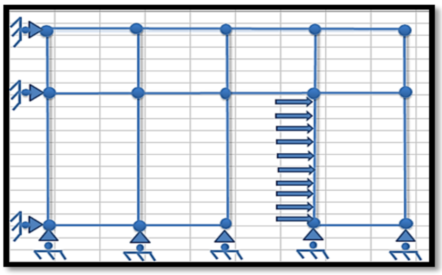


Fig.1 g regular model of tongue with uniform presure with boundary conditions

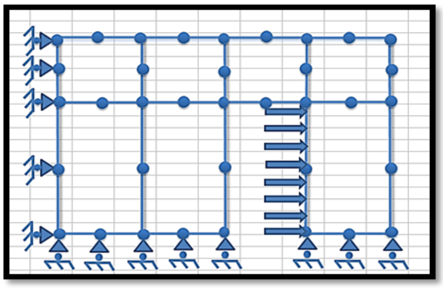


Fig.1 h regular model of tongue with uniform presure with boundary conditions

The seat belt tongue is subjected to 1000 lbf tension load. For analysis purpose that tension is converted into uniform pressure of about 14000 psi along the 0.75 inch vertical inside edge of the latch retention slot.

The basic steps which included in calculation are as mentioned below:

Step-0: Solution Strategy.
Step-1: Generation of an equilibrium matrix for the system.
Step-2: Generate the compatibility conditions.
Step-3: Generation of flexibility matrix for the system.
Step-4: Express the compatibility conditions in terms of forces.
Step-5: Couple the equilibrium equations and compatibility conditions to obtain the IFM equations, and solve for the forces.
Step-6: Back calculate the displacements, if required, from the Forces.

Looking to the problem by coarser discretization, seven nos. of rectangular elements are generated. Generation of equilibrium equations for the seven elements of seat belt component proceed by standard assembly procedure. The system equilibrium matrix [B] of dimension (22 × 35) and (59 × 91) for REC_5F_8D and REC_13F_16D elements respectively are generated. As dimension of the global equilibrium matrix shows sparse matrix, to make it square matrix CCs are generated from the elimination of DDRs. The global flexibility matrix for the problem is obtained by diagonal concatenation of the seven elemental flexibility matrices of dimension (35 × 35) and (91 × 91) for REC_5F_8D and REC_13F_16D respectively.

Von mises stress calculation is the extension part of the results at notch which situated on right-hand edge of the latch retention slot.

Von Mises stress for a two–dimensional plane stress state (σ3 = 0), can be stated as in following equation:

$$\sigma_{vm} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 - \sigma_{xx}\sigma_{yy} + 3T_{xy}^2} \quad (3)$$

The procedure for the von mises stress which is an average value, at the notch is counted for the element 3, 4 and 5 as follows and also representation of it as depicted in Fig.1. i.

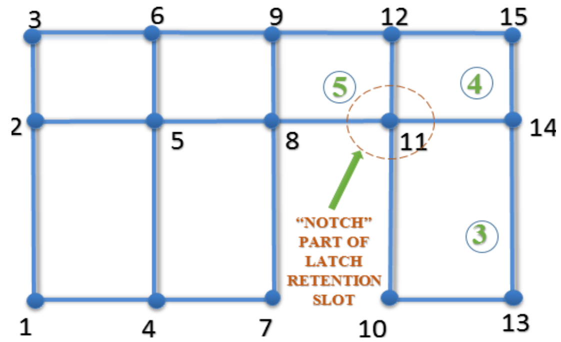


Fig.1 i Von mises stress location at latch retention slot

Individual force values as per element types (i.e. REC_5F_8D and REC_13F_16D) are worked out from above procedure of IFM. Von mises stress are calculated at junction of the three different elements. The purpose of failure theories is to extend the strength values obtained from uniaxial tests to multi-axial states of stress which exists in practical existing engineering structures.

V. RESULTS AND DISCUSSION

The maximum displacement in pull direction is about **0.0016 inches** which occurs at expected at the center of the right-hand edge of the latch retention slot for both types of rectangular plane stress FE elements. The von mises stress value is **98, 067.31 psi** and **1, 13, 430 psi** for REC_5F_8D and REC_13F_16D respectively using IFM. For better understanding of IFM formulation results it can be compared with ANSYS (Workbench 14.5). Result obtained for maximum displacement in Pull direction by the ANSYS (Workbench 14.5) is **0.00189 inches** and von mises stress value is **73,795.201 psi (Regular Coarser meshing).**

VI. CONCLUSION

The IFM always exhibits good performances for better stresses and nodal displacement with coarser discretization. IFM provides a strong motivation to reexamine the relative merits of the force and displacements methods within the context of the finite element idealization. Procedure wise IFM and

FEM are seems to the same but IFM proves to be more correct as error due to differentiation of displacement polynomial is totally removed which is available in FEM procedure.

REFERENCES

[1] S. N. Patnaik, D. A. Hopkins and Halford, *Integrated force method solution to indeterminate structural mechanics problems, NASA/TP—2004-207430*

[2] I. Kaljevic, S. N. Patnaik and D. A. Hopkins, *Development of finite elements for two-dimensional structural analysis using the integrated force method, Computers & Structures, Vol. 59, No.4, pp.691-706, 1996*

[3] S. N. Patnaik and M. S. Nagraj, *Analysis of continuum by the integrated force method, Computers & Structures, Vol. 26, No.6, pp.899-905, 1987*

[4] S. N. Patnaik, R. M. Cornoneous and D. A. Hopkins, *Recent Advances in the method of forces: integrated force method of structural analysis, Advances in engineering Software, Vol. 29, No. 3-6, pp. 463-474, 1998*

[5] S. N. Patnaik and K. T. Joseph, *Generation of the compatibility matrix in the integrated force method*, Computer methods in applied mechanics and engineering, 55, (1986)