THREE DIMENSIONAL FINITE ELEMENT MODELLING OF BEAM TO COLUMN END PLATE STEEL CONNECTION

Hasmit Prajapati[1]

[1]*Applied Mechanics Department, Faculty of Technology and Engineering, The M. S. University of Baroda.*

Abstract —*This paper presents three dimensional finite element model for End plate steel connection using ANSYS software to obtain it's rotational behavior and Stress distribution in each of its elements. In this model geometrical non linarites, Bolt pretension, Contact simulation and material nonlinearities are simulated. For understanding stress distribution in connection a design example as per IS: 800:2007 is modeled in ANSYS.*

Keywords- *ANSYS, Beam-Column Connection, Finite Element Analysis, M-θ Curve.*

I. INTRODUCTION

Steel portal frames are traditionally designed assuming that beam-to-column joints are ideally pinned or fully rigid. This simplifies the analysis and structural design processes, but at the expense of not obtaining a detailed understanding of the behaviour of the joints, which have a finite stiffness and are semi-rigid. The last century saw the evolution of analysis methods of semi-rigid joints, from the slope deflection equations and moment distribution method to the matrix stiffness method and nowadays iterative method coupling the global structural analysis with the analysis of the joint. Studies reveal that in frame analysis, the joint rotational behaviour should be considered. This is commonly expressed by using the moment rotation curve. Availability of high speed computers and sophisticated FE analysis software encouraged the researcher to address the steel-connection problem using computers rather than expensive experimental work. The present work is aimed at developing full 3D FE model of End plate beam column steel connection using ANSYS v.16 to obtain moment rotation curve and stress distribution in each elements while comparing same with design example as per IS: 800:2007 . This model includes geometrical and material non-linearity, contact simulation and bolt pretension.

II. PROBLEM DESCRIPTION

End plate connection is considered one of the stiff beam to column connection as in this connection beam and column are connected by end plate where beam is welded on end plate as shown n in Figure 1. Details of the connection elements are presented in Table 1.

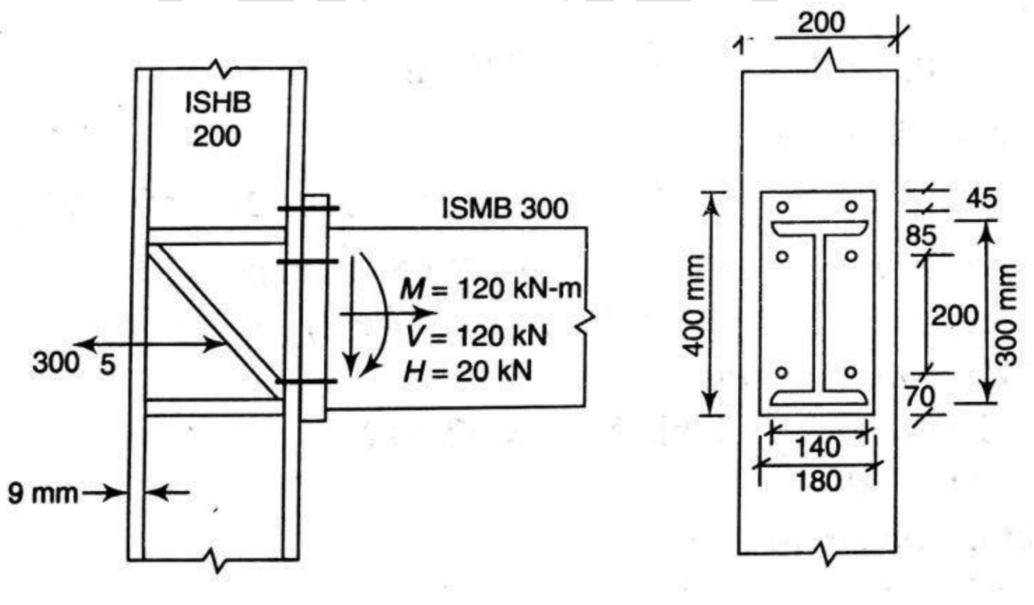

Figure 1 End plate Connection detail

Table 1 End plate Connection details

Sr. No	Elements	Standard	Section
1	Column	INDIAN	ISHB 200
2	Beam	INDIAN	ISMB 300
3	End plate	INDIAN	20mm thick
4	Weld	Machine weld	6m
5	Bolt	INDIAN (HSFG BOLT)	20mm (Dia.)

III. CONSTRUCTION OF FE MODEL

The FEM model is constructed with the following particulars:

i. Diameter of bolt and hole is same, thus there is no gap between hole and bolt to prevent initial slip.
ii. 3D geometry of connections are created in AutoCAD and then imported in ANSYS as shown in Figure 2.
ii. All the contacts in model are considered as frictional contact with co-efficient of friction as 0.4.
iii. Solid bolt with same head at top and bottom is simulated as nut and bolt.
iv. Programme controlled meshing is chosen with medium mesh option.
v. Discretization of model is done by four noded brick elements and four noded tetrahedrons as shown in Figure 3.

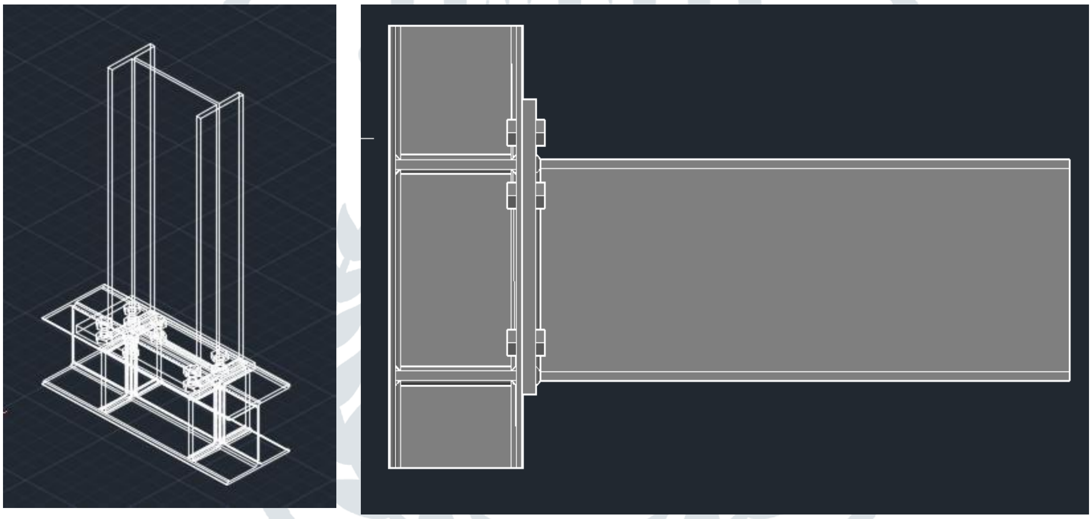

Figure2 Geometry of End plate Connection in Autocad

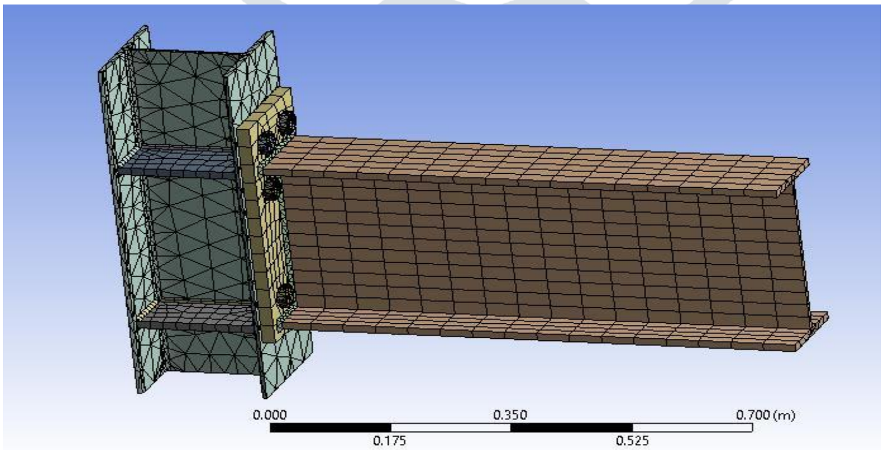

Figure3 Discretized model in ANSYS

IV. MOMENT ROTATION (M - θ) CURVE FOR END PLATE CONNECTION

For generating moment rotation curve for connection moment is applied at free end rotation at connection is measured. In Figure 4, boundary conditions and loading considered in the model are represented. Fixed support condition at top and bottom of the column is assigned. Moment is applied at the end of the beam and for simulation of initial force developed in bolt due tightening of nut, additional bolt pretension force is applied. It is calculated using empirical formula.

P = (0.8) fy * Ab

Where, Ab is cross-section area of bolt and fy is its yield strength.

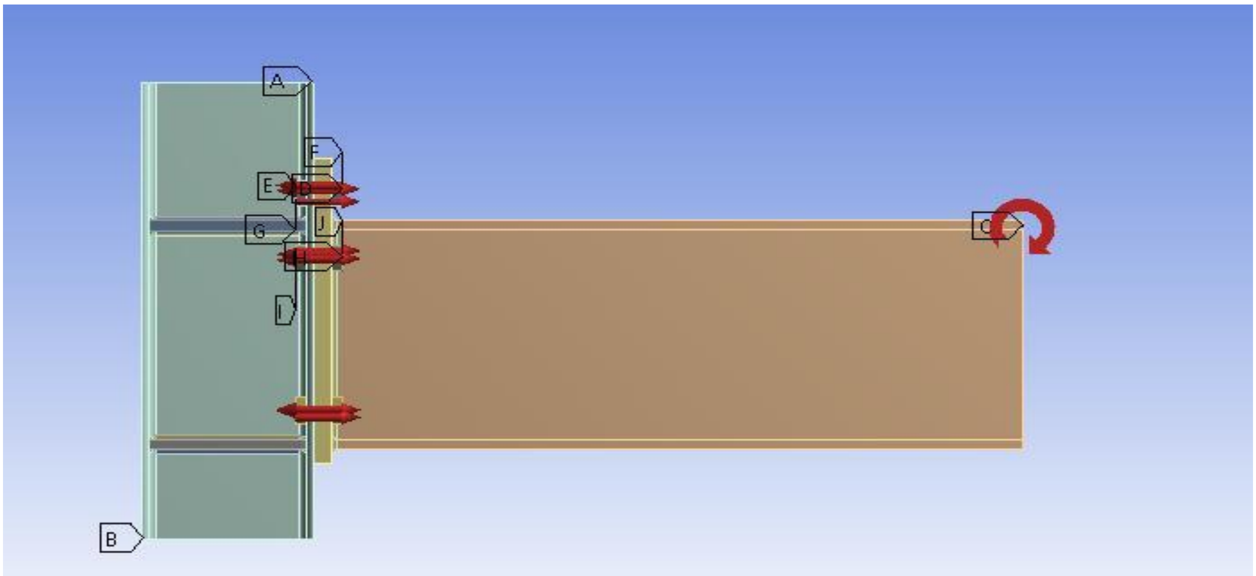

Figure 4 Applied Boundary Condition

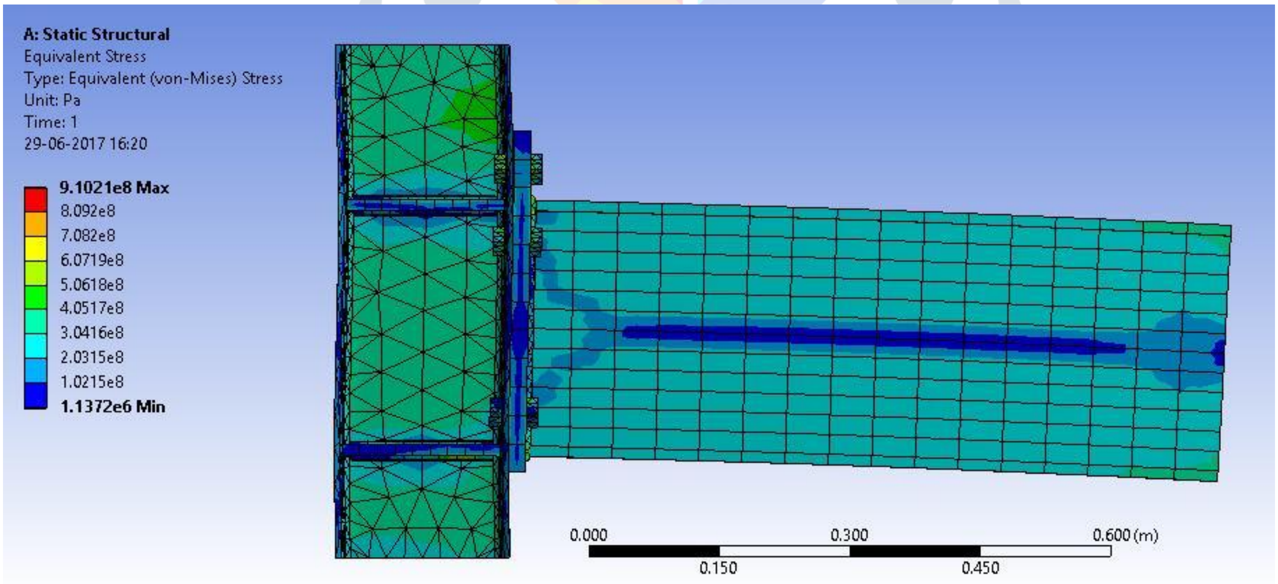

Figure 5 Von-misses stresses for Connection

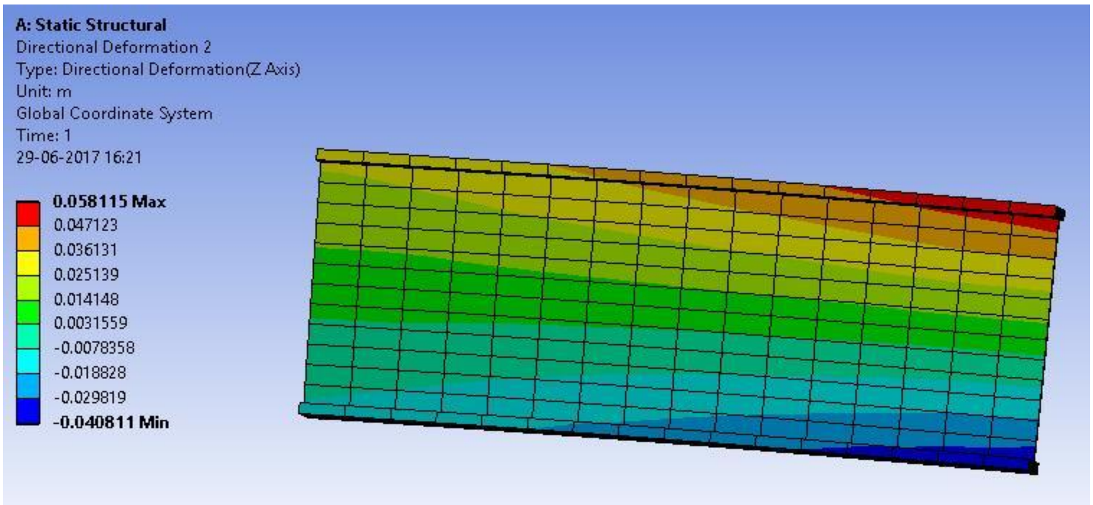

Figure 6 Directional deformation of connection

Figure 5 represents pictorial view of distribution of stresses due to application of moment at free end. It is clearly seen that minimum stresses are at free end and maximum stresses are at joint. Figure 6 shows directional deformation contour for the beam which shows maximum deflection at the free end as expected.

Table 2 Moment-Rotation results

Sr.NO	Moment kN.m	Deformation mm	Rotation (ANSYS) rad.	Rotation (ANSYS) (x 10^-3) rad.
1	0	0	0	0
2	5	0.06	0.00040	0.40
3	10	0.12	0.00080	0.80
4	15	0.18	0.00120	1.20
5	20	0.24	0.00160	1.60
6	25	0.3	0.00200	2.00
7	30	0.37	0.00247	2.47
8	35	0.42	0.00280	2.80
9	40	0.48	0.00320	3.20
10	45	0.54	0.00360	3.60
11	50	0.6	0.00400	4.00
12	70	0.85	0.00567	5.67
13	100	2.0	0.01333	13.33
14	150	9.69	0.06451	64.51
15	200	47.06	0.30401	304.01

Table 2 presents results obtained from ANSYS for moment versus rotation and Fig. 7 shows graphical representation of the same.

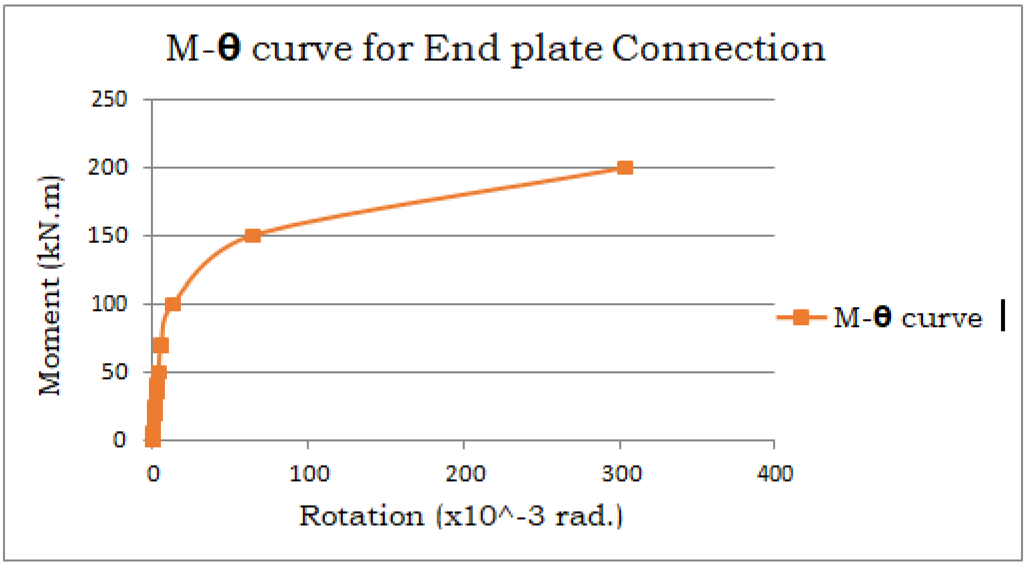

Figure 7 M-θ curve for End plate connection

As known typical M-θ curve has two parts one is linear and another is non- linear. The point at which curve changes from linear to non-linear should be considered as yield point. Here in this case graph and table shows that at 70 kN.m connection starts showing non-linear behavior. So the curve from 0 kN.m to 70 kN.m is linear and slope of this part will give initial rotational stiffness of connection which is found to be 12500 kN.m/.rad.

V. STRESS ANALYSIS OF CONNECTION

For stress analysis same connection as shown in Figure 1 and Table 1 is considered. Design of this connection is carried out as per IS: 800: 2007. Which suggests that Shear capacity of this connection is 120 kN and moment capacity is around 120 kN.m. Same boundary condition is applied in ANSYS model which is shown in Figure 8. Shear force is applied in form of pressure on top of end plate to avoid any local yielding due to point load.

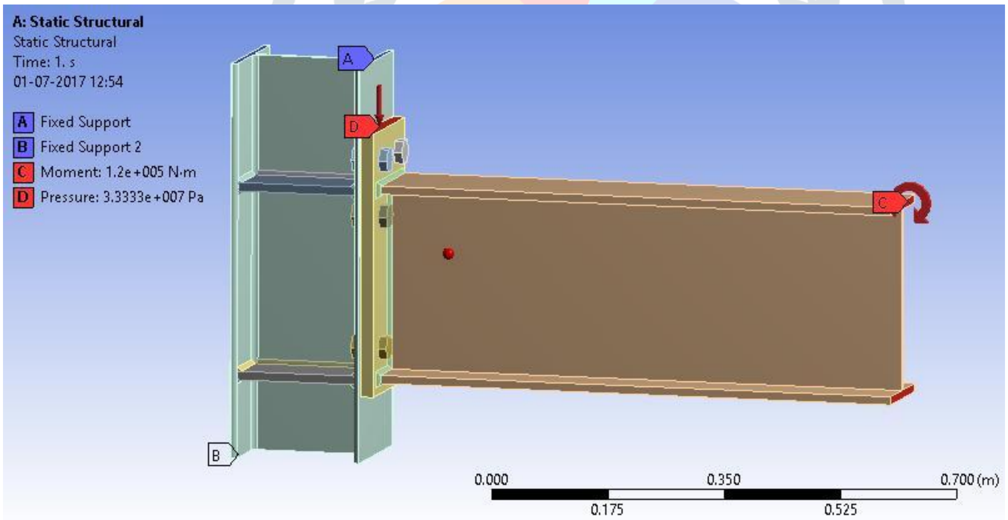

Figure 8 Boundary condition for Stress analysis

Figure 9 shows von – mises stresses generated in connection which are later on used for determination of safety of factor of connection. Figure 10 shows stress ratio for the connection which represents nearly same contours as represented in case of von-mises stresses as at free end of beam stress ratio is less and at connection region it is around 0.75 to 0.85 which can be considered as safe condition. Figure 11 represents FOS for connection as per Distortion energy failure theory. This results from ANSYS shows that connection is in safe condition as FOS contours are above one for the entire connection.

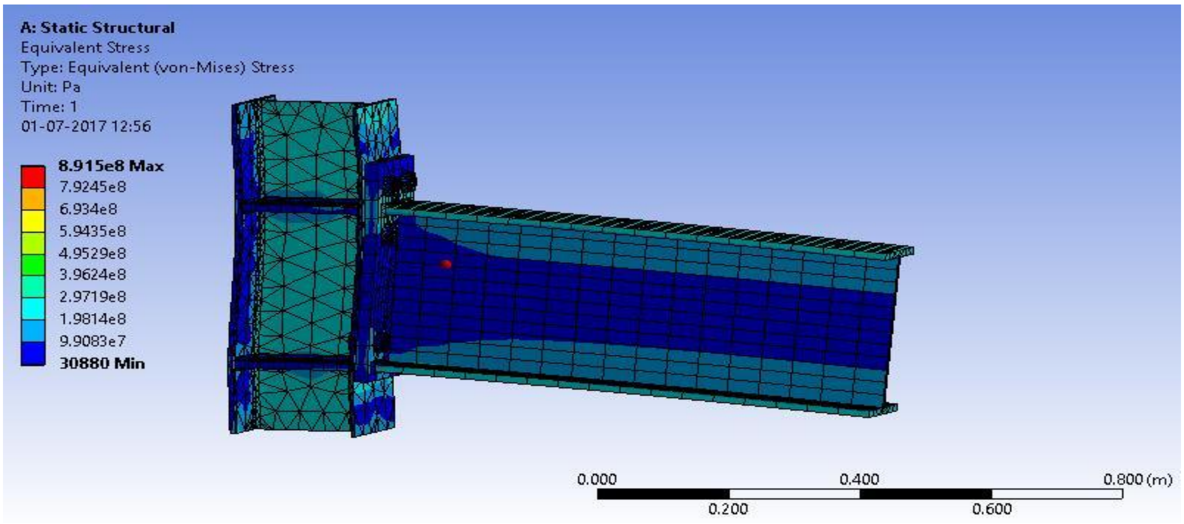

Figure 9 Von-misses stresses for connection

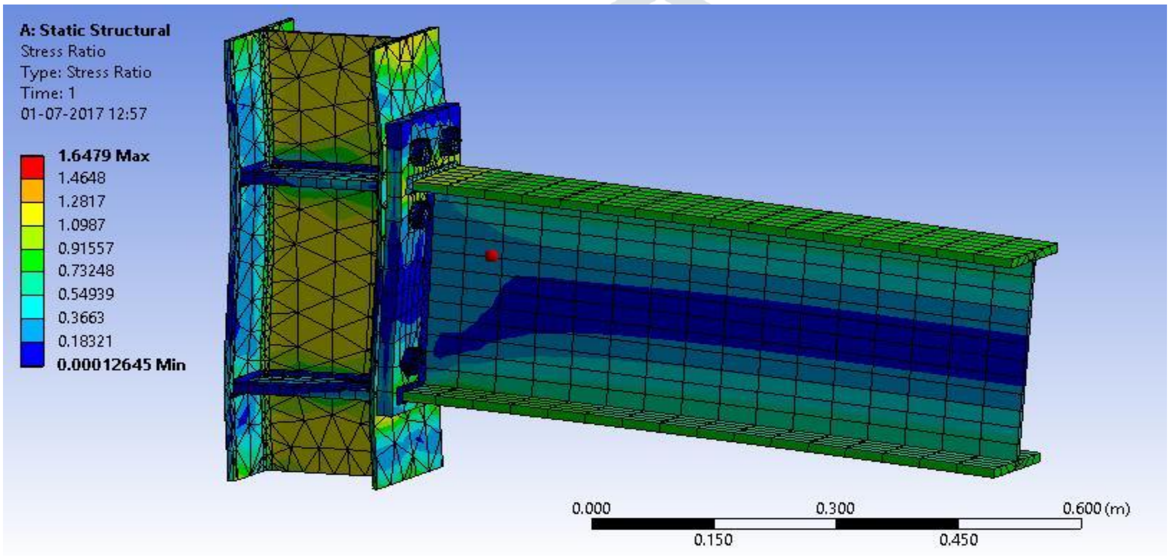

Figure 10 Stress ratios for connection

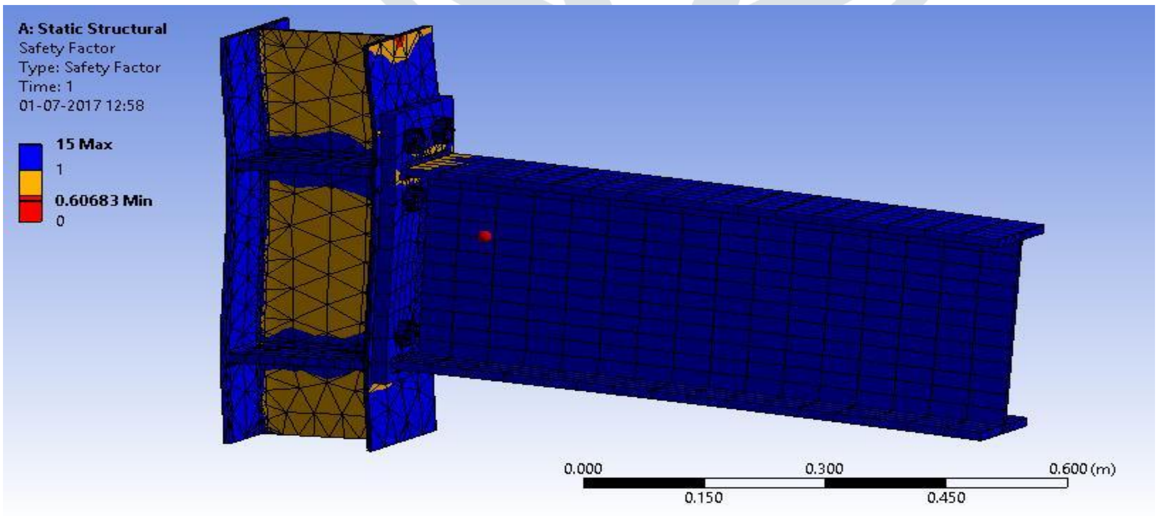

Figure 11 Factor of safety for connection

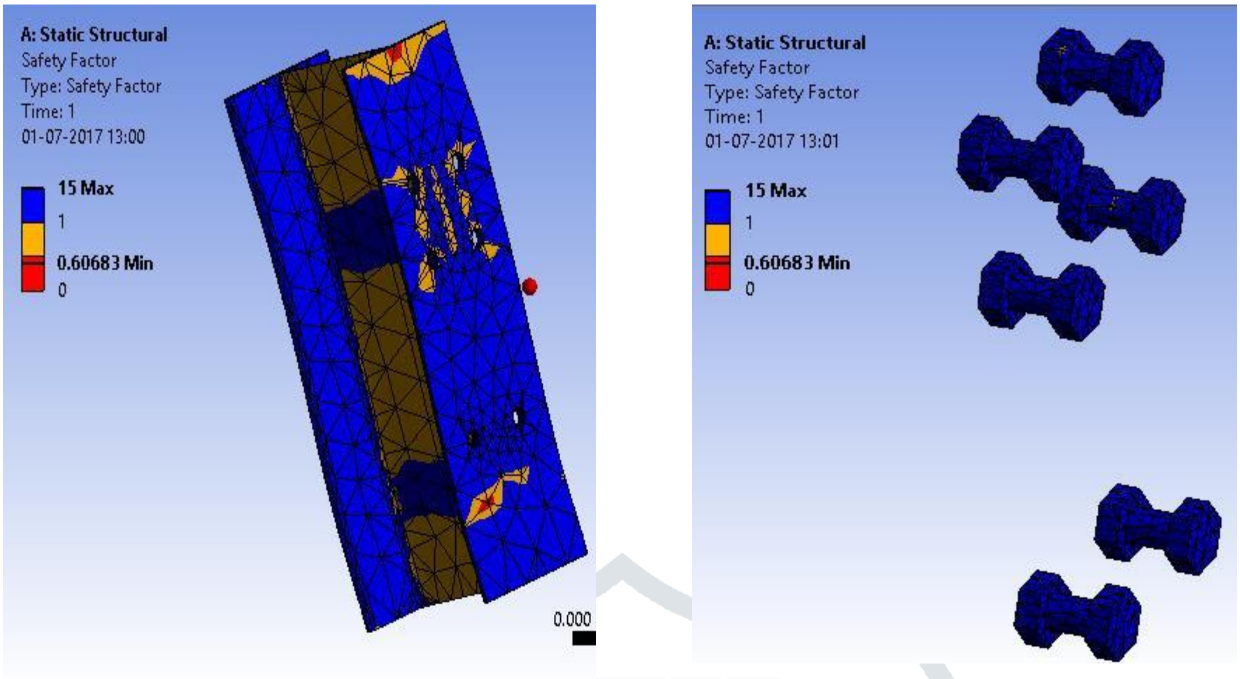

Figure 12 Factor of safety for Bolts and Column

Figure 12 shows FOS for column and bolts from which one can say that some edges of bolts are having FOS around 0.9 to 1.0 and on some portion of column flange FOS is around 0.6 to 1.0 which shows local yielding of column web and flange which is quite obvious as column is weak in comparison of beam.

VI. CONCLUSIONS

From results it can be clearly understood that ANSYS is very good tool to simulate behavior of structural steel connection.

Profile of Moment rotation curve obtained for End plate connection matches with profile of typical moment rotation curve in which it is divided into two parts. One is linear part which gives initial rotational stiffness of connection and second is non-linear part which represents non-linear behavior of connection.

Initial rotational stiffness of this connection is found to be 12500 kN.m/.rad. So this connection can be considered as stiff connection.

Moment and Shear capacity of this connection is found to 120kN and 120kN.m. Same has been modeled in ANSYS and it is found that that connection is safe as per distortion energy failure theory.

In results provided by ANSYS it is observed that stresses near top of end plates are higher, which could be due to load is not applied at center of connection.

ANSYS also simulates location of minor yielding which is not there in case of design procedure. Thus ANSYS provided very good stress distribution and can easily handle any complex geometry.

VII. REFERENCES

1. N. Krishnamurty "Correlation Between 2- and 3-dimensional Finite Element Analyses of Steel Bolted Endplate Connections" *Computers & Structures*, Vol.6, pg. 381-389, 1975.

2. A. Azizinamini "Initial Stiffness of Semi-Rigid Steel Beam-to-Column Connections" *J. Construct. Steel Research* (1987) 71-90.

3. J.-G. Yang "Three-dimensional Finite Element Analysis of Double Angle Connections Under Tension and Shear" *J. Construct. Steel Research* 54 (2000) 227–244.

4. Concepción Diaz "FEM model of Beam-to-Column Extended End-plate Joints*" J. of Constructional Steel Research* 67 (2011) 1578–1590.

5. Mohammad Javad “A new Model for Beam Rigid Connection to Double I-shaped Column's Web” *J. of Constructional Steel Research* 127 (2016) 204–220

6. Azizinamini A. “Cyclic characteristics of bolted semi-rigid steel beam to column connections.” *PhD thesis, University of South Carolina*, Columbia, 1985

7. Software Manual of ANSYS v16.0

8. IS: 800:2007 “GENERAL CONSTRUCTION IN STEEL— CODE OF PRACTICE by Limit State