

Finite Element Analysis of Steel Square Hollow Beam connected to Square Hollow Column Join

Nilesh Kumar¹, Chandrashekhar kurre², Mrs. Gargi Danda De³

¹Department of Civil Engineering, SRU, Raipur, Chhattisgarh

²Assistant Professor, School of Engineering & Tech., ISBM University, Chhura

³Assistant Professor, Department of Civil Engineering, SRU, Raipur, Chhattisgarh

Abstract — In India, standard steel sections are used to make the majority of steel structures (such as angle, channel and beam sections). However, new hollow steel sections (like square and rectangular hollow sections) are becoming more and more popular in modern steel constructions because of a number of benefits including a higher strength to weight ratio, better fire resistance properties, a higher gyro radius, a smaller surface area, etc. Compared to ordinary steel sections, these hollow sections can reduce costs by between 30 and 50 percent (Tata Steel brochure, 2012). However, unlike ordinary steel sections, these hollow sections lack standard connection specifications that can be found in public literature or design codes. The goal of the current study was to create a solution to this dilemma.

A square hollow beam to square hollow column connection was chosen and modelled in the commercial finite element programme ANSYS to meet the aforementioned goal. In light of various connection characteristics, this model was examined for nonlinear static (pushover) analysis. For this investigation, the following four alternative connection scheme details were chosen: (i) channel-1 basic structure, (ii) channel-2 using end plate (iii) channel-3 using closed structure (iv) channel-4 T-shape section. For comparison, the base model, which consists of a rectangular hollow beam welded to one face of a rectangular hollow column, is also looked at. For joints between rectangular hollow beams and rectangular hollow columns, the performance of the chosen connection details is compared, and the best performing connection details are advised.

The outcome demonstrates that the type of connection employed has a significant impact on the joint's maximum deformation capacity and load carrying capability. Additionally, the type of connection utilised has a significant impact on where in the structure plastic hinges form (which can be at a joint or a beam).

Index Terms— analysis of hollow section, 3D modelling, finite element analysis, CAD analysis, structural analysis.

I. INTRODUCTION

The majority of steel buildings in India are constructed using standard steel sections (such as angle, channel, and beam sections) and were created using standard working stress techniques. However, due to a number of benefits, new hollow steel sections (such as square and rectangular hollow sections) are becoming more and more prevalent in contemporary steel constructions. Because they have a high strength to weight ratio and are more effective at resisting forces than conventional steel sections, hollow steel sections not only make the total structure lighter.

Additionally, the hollow parts are more fire resistant. Transport, manufacture, and painting expenses can be reduced by increasing the gyrating radius and decreasing the surface area of the sections. A recent study (Tata Steel Hollow section brochure) claims that adopting hollow steel sections rather than conventional steel sections can result in cost savings of up to 30 to 50 percent. Such hollow sections were first produced by the Indian steel industry and regularly made available to the construction industry. This trend has drawn experts' attention to the connection design, a crucial component of steel design and construction. These hollow steel sections do not have standard connection specifications that are listed in design codes or published literature, unlike normal steel sections. Because acceptable connection configurations (for shear force and bending moment) have not been developed between the sections, the application of hollow sections is not growing.

There is enough information available about the seismic connections between conventional beams and columns, particularly I-beams and I-columns, but relatively little about the seismic connections between

hollow beams and hollow columns. Given that the geometry of hollow and conventional sections differs in some ways, direct expansions from connections detailed between the conventional sections are also impractical. Hollow box sections often have very thin walls (4–8 mm thick), so if a connection is constructed without sufficient analysis, the likelihood of local failure is very high.

The standard procedure is to introduce diaphragms at beam flange level into columns for large hollow sections and to employ full penetration welds for junctions involving smaller hollow sections. However, this requires a lot of fabrication effort and is only partially capable of preventing fracture under seismic loads, driving up the price of the construction. Bolts can be used as an alternative since they allow for significant manufacturing tolerances, but they cannot always be used everywhere because traditional bolts need access from the interior to be tightened. To solve these issues, a suitable and affordable connection detail between two hollow pieces that is both simple to construct and strong enough to withstand loading should be created. This is the main driving force behind the current investigation.

FINITE ELEMENT ANALYSIS

The extensive selection of elements in the ANSYS 19R3 element library allows for modelling freedom for various geometries and architectures. The following factors can be used to characterise each element: Family, Number of nodes, Degrees of freedom, Formulation and Integration.

ELEMENTS USED FOR MODELLING

The main elements used for modelling RC beams are beam elements, bar elements, and shell elements. There are various kinds of beam elements in the ANSYS 19R3 element library. The main solution variables are merely functions of position along the beam axis thanks to an element known as a "beam" that makes mathematical assumptions that simplify the problem to one dimension (as bar element). The shortest distance from an axis defined in a beam to any point in the continuum must be minimal compared to normal lengths along the axis.

II.METHODOLOGY

Finite Elements Analysis

It is the advanced technique to find out the optimum solutions and simulation of structural problems by using different computer software. This type of software has the capacity to analysis very difficult numerical mathematics to find out the accurate results by taking a while. It can also generate the graphical and simulation representation on the basis of various parameters.

The following methodology has been developed to attain the aforementioned goals:

- (i) Determine the geometry of the connection between a square hollow beam and a square hollow column.
- (ii) Use the commercial finite element programme ANSYS 19R3 to simulate the chosen connection.
- (iii) Create a plan for any potential alternate beam-to-column connection information.
- (iv) Conduct a nonlinear static analysis of the chosen beam-to-column connection while taking into account all of the chosen connection information.
- (v) Determine the best connection detail for the chosen beam-to-column connection by taking into account the flow of loads or forces, the joint's ability to support its weight, and the resulting ductility.

Static analysis of structure FEA Approach

This chapter explains the approach used for the static finite element analysis of the backhoe excavator's rectangular steel channel and rectangular steel channel in ANSYS. Additionally, boundary conditions, mesh models, and findings are shown and discussed, along with rectangular steel channel teeth. By taking into account the determined maximum breakout force configuration, FEA is performed on two components of the backhoe excavator attachment.

FEA procedure in ANSYS

The steps below are used to analyse the entire rectangular steel channel assembly of the three-dimensionally modelled ANSYS backhoe excavator attachment: -

1. A dialogue box A with engineering data, geometry, model, setup, solution, and results will show on the project schematic screen when ANSYS 19R3 is opened and static structural is selected from the analysis system list. When you right-click on geometry, you can import geometry and browse through models that have already been made in Creo 3.0 in .stp file format. If this geometry is supported by Ansys, a right mark will show up when you open it,

choose the unit, and click Generate. The model will then display on the screen. Select the close design module under files.

2. Describe the materials: For a simulation, material attributes define the structural qualities of each component of the assembly (FEA). Additionally, ANSYS enables a user to alter the necessary material properties by selecting add new material and specifying their properties that need to be analysis of the parts in each simulation, conducting various outcomes. Click on update project and then go back to the project after choosing the necessary material.

3. Open the project schematic model. A 3D model will emerge on the screen so you can thoroughly inspect the design. The project's geometry, coordinate system, static structure, and solution are all presented on the right side of the window beneath the outline. To view all the information of this geometry area, including the dimensions of the moment of inertia, first choose the geometry and then the model's name. We must assign the material that was developed in the engineering data part in the material section.

4. Specify contact conditions. In ANSYS, there are two ways to add contact conditions: Automatic contacts and Manual contacts. While manual contacts provide the user more freedom to choose the contacts and allow for post-assignment editing, automatic contacts software assigns the contacts based on the parameters in the edit simulation properties dialogue box.

5. Meshing: If the required number of nodes and elements appear in the static area, right-click on the mesh and choose Generate Mesh, Edit Elements Size. In general, geometric integrity is a requirement of design analysis tools. These integrity standards might not be met or might even go unrecognised during the CAD modelling process. Small gaps, overlaps, or overhangs, which are simple to miss from a CAD perspective, are examples of integrity problems. However, they can cause mesh a lot of problems. In order to prevent or suppress sections that have extremely small features relative to the overall model dimensions and do not significantly affect the simulation results, geometry clean-up must be completed prior to meshing.

6. Include restrictions: Structural constraints limit or restrict the model's displacement. To use any part or assembly as an example for static simulation, all stiff

motions must be eliminated (all translational and rotational movement of the body).

7. Boundary conditions: Apply various loads to the body by right-clicking static structural. Structure loads are the pressures placed on a component or assembly when it is in use. Components under such loads experience stresses, deformations, and displacements. Knowing how the component responds in both typical and extreme working situations is crucial while developing the component. Force (N), pressure (MPa), hydrostatic pressure, remote force, bearing load, moment, and linear pressure are the several load types that are supported by ANSYS.

8. Solution: After selecting solve from the context menu for the solution, all the parameters and boundary conditions that were added to the model will need to be solved. It will display the solution file if all of the supplied information is accurate. Select total deformation, equivalent stress (von misses), etc. by right-clicking the answer. Select review all outcomes. The findings will now be displayed, together with the crucial information. Additionally, it has a word file save option after creating a project report.

FEA of steel structure

To determine whether the produced stresses fall within the safe stress limit of the material used for structural analysis, static FEA is being performed in this case. This analysis can be used to determine.

The static finite element analysis (FEA) of the various structures used in the proposed study applies boundary conditions computed using the ANSYS workbench software. Tetrahedral elements are created by the software itself. fine meshing is used, which produces correct findings, taking into account the material qualities as stated. The post processing takes into account the stresses and displacements produced by Von Misses when known boundary conditions are applied.

Since the study's materials for the steel structure is explained, Von Misses stresses operating on the parts should be taken into consideration when designing all of the parts. Von Misses stresses are utilised as a criterion in determining the onset of failure in ductile materials.

The load placed on the backhoe pieces causes them to deform. When analysing the backhoe attachment parts for safe stress conditions, the deformation or displacement of the backhoe part should be less than the minimum thickness of plate utilised in those parts. The FE analysis of the rectangular steel channel is presented in the following section.

Material for structure

When a structure is subjected to significant deformations, the results of linear elastic analysis do not accurately describe the behaviour of the structure under ultimate loads (due to forces like earthquake). The stress-strain curve of steel is taken into consideration in the current study to model the material nonlinearity. The stress-strain relationship of the mild steel employed in this study. It is believed that steel has a typical strength of 250 MPa in both tension and compression.

Material properties should be defined for the steel structure before modelling. Table no 2.1 showing below the mechanical properties of structural steel on the basis of different parameters. In ANSYS software material properties defines before the modelling of the structures.

Table 2.1 Mechanical properties of Structural steel

Sr.no.	property	Value	Unit
1	Density	7850	kg/m ³
2	Modulus of elasticity	200000	Mpa
3	Poisson's ratio	0.3	
4	Yield strength	250	Mpa
5	Ultimate tensile strength	410	Mpa

Modelling

The responses of beam-to-column junctions can be studied using a variety of techniques. Of all possible ways, experimental testing would be the most successful. However, both in terms of time and money, experimental procedures are exceedingly expensive. Therefore, in the current work, the joint was analysed using the finite element method. Due to improved knowledge and capabilities of computer software and hardware, finite element analysis has become more prevalent recently. Modelling components with

computer software is significantly faster and more economical. By analysing the structures using finite element analysis, reasonable conclusions can be reached.

The quality of the results depends on the precision of the modelling, which is a crucial component of finite element analysis. It is required to understand the element type as well as its characteristics and behaviours in order to model and analyse a beam column junction comprising two box sections. Understanding the various types of elements utilised in the FE software ANSYS 19R3 is also crucial. There should be a clear understanding of which ANSYS 19R3 element is better for 2D approximation and which is better for 3D approximation. The material modelling employed in the current investigation is covered in this chapter's discussion of finite element modelling.

GEOMETRIC MODELLING

As the components needed for modelling in CAD software have already been covered in the paragraphs above, the modelling of the problem's geometry is now a crucial step in moving this research issue forward. As it has already been said that the primary goal of this job is to build a suitable connection between two box sections. This section includes the many sorts of suggested connection details that were modelled using Creo 3.0, as well as their physical and FE characteristics and boundary conditions. In CREO PARAMETRIC 3.0, an extrusion was used to obtain the geometry of the different Structures. There is total four different structures needed to create in Creo 3.0 software.

Channel-1 Basic Connection Structure

Two exactly similar square hollow sections are modelled with the help of Creo- Para metrics 3.0. No welding or bolting is done in this connection. The dimension of both the sections are 200×200×8 mm. Length of both the sections are 3000 mm. Boundary conditions are kept at the column ends as both sides hinged. The connection is shown in the below. Figure no. 2.1 showing channel-1 basic connection structure

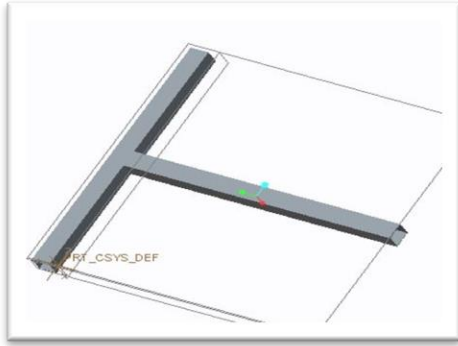


Figure No. 2.1 Basic connection detail of channel- 1 structure

CHANNEL - 2 CONNECTION DETAILS USING END-PLATE

The beam column joint of the fundamental connection, which is depicted above, has an end plate linked to it. The end plate has a thickness of 8 mm. The end plate's mesh is maintained at a length of 700 mm square. The plate measures 200 x 1000 x 8. Figure no 3.3 shown below.

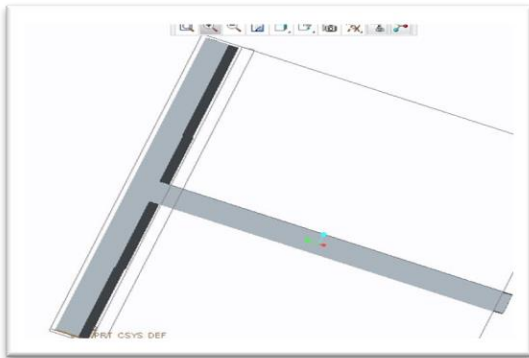


Figure No. 2.2 Channel -2 Connection using End-Plate

Channel – 3 Connection Detail using closed channel Section

At the intersection of the beam and column, four angle sections are connected with box sections using this style of connection detail. Angle section mesh sizes are maintained at a length of 700 mm square. Each angle portion has the following measurements: 216x216x8 mm. Figure No. 2.3 shown below.

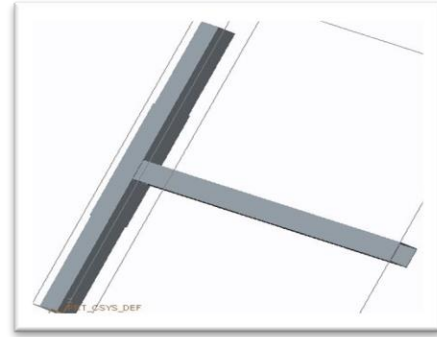


Figure No. 2.3 Channel – 3 Connection detail using closed channel section

Channel – 4 Connection Detail using T-shape Sections

This style of connector details employs two channel portions. They mostly serve to cover the intersection of a beam and a column. Each channel portion has a dimension of 216x 600x 8 mm. The channel segment has a 80 mm square mesh size. Figure no. 2.4 shown below.

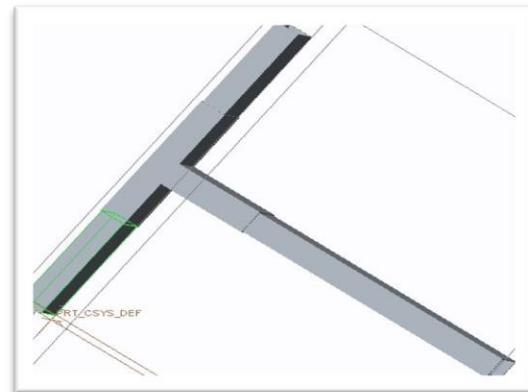


Figure No. 2.4 Channel – 4 Connection Details with T-shape section.

Boundary conditions and analysis of Channel-1 Basic Connection Structure

Finite elements analysis needs to be done in ANSYS 19R3 software. First geometry is imported in model section then assign materials properties used for the rectangular steel channel. Properties of the materials used in rectangular steel channel is listed in table 2.1. All the parts of structures of the rectangular steel channel are made up of HARDOX 410 Mpa(Yield strength, $\sigma_y = 250$ MPa). Fig. 2.6 shows the boundary conditions applied to rectangular steel channel as

calculated for maximum breakout force configuration. the material properties of the given structure. After assign the material properties of the structure the next process is meshing it is the important part of the finite elements analysis to calculate the accurate and optimum result of the structure. Fig. 2.5 shows the mesh view of the arm with 3221 nodes and 4526 elements. After meshing of the structure boundary condition should be define and fixed for applying load on the structures. Applying fixed support on the structure is shown in below figure no. 2.6 Fixed Support of Channel -1 structure flangs type. Now the boundaries of the structure are fixed and the next step is to applying the forces in various scales like 50 KN, 100KN, 200 KN, 300 KN, 400 KN and 500 KN. These forces are applied on the upper portion of horizontal beam of the structure as shown in figure no. 2.7.

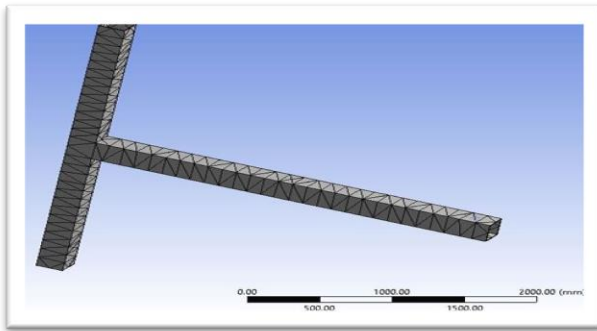


Figure No. – 2.5 Mesh view of the channel -1 fanggs type

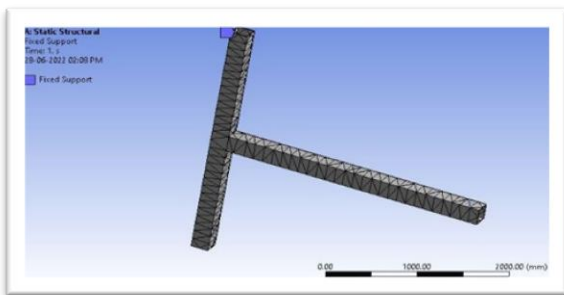


Figure No. 2.6 Boundary condition and Fixed Support of Channel -1 structure flangs type

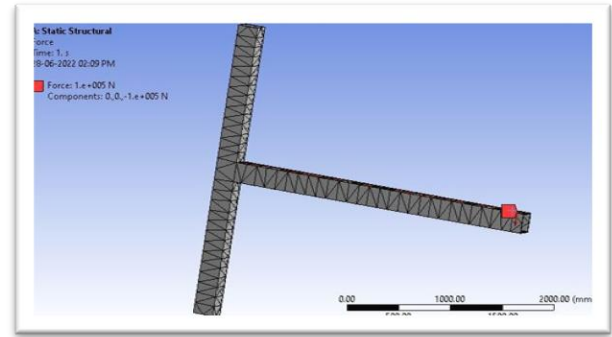


Figure No. 2.7 100 KN force applied on Fixed Support Channel -1

This method of analysis needed to performed as channel-1 for all the proposed design of structures.

III RESULTS

The primary goal of this study project is to create a proper connection between two box sections while taking welding connectives into consideration, as was stated in the previous chapter. The first thing to take into account is the parameters needed to complete this work in order to attain this goal. The thorough literature review that was conducted for this research project revealed certain inherent challenges in creating a good connection between two empty portions. The primary challenge is that past works have primarily been between one hollow section and one conventional piece.

Therefore, due to geometrical discrepancies, straight extensions from those connection details are not possible. Some new criteria need also be taken into account for this. So, using ANSYS software, a FEA analysis is first performed in this chapter for the model consisting of two Square Hollow sections up to failure. The essential characteristics, such as force flow and the site where a plastic hinge forms, are then sorted out by analysis of the data, and some suggested connection details have been developed to meet the requirements. The proposed connection details were then thoroughly compared in order to choose the one that met the problem's requirements in every way.

Therefore, this chapter focuses on modelling, analysis, and selection of the best connection detail among those proposed that meet the requirements for constructing a good connection detail.

Channel -1 BASIC CONNECTION DETAIL

Two appropriate hollow sections have been selected from the Indian Steel table in order to move forward with the current study project. In this instance, two identical square hollow parts are picked. The parts are 200x200x8 mm in size. Both parts' lengths are taken to be 3000 mm. Mesh sizes and the modelling component were discussed in the prior chapter. The parts are then put together to form an abeam column construction. The boundary conditions at the column's two ends are maintained as hinged, and load is delivered to the beam's tip.

Applying boundary conditions, loads and meshing

The structure is analyzed using ANSYS software while applying the load to the tip of the beam until failure occurs. According to the results of the analysis, the structure's beam-column joint, which is depicted in figure 3.2, is where the majority of the stress is concentrated. This suggests that a plastic hinge has formed at the column, which is bad for a seismic connection. The plastic hinge should form far from the joint of the beam and column for a seismic connection. The maximum principal force flows from the beam centerline to the column web, which is the analysis's other key finding. Figure 3.1 shown below total deformation and figure 3.3 shown maximum principal stress.

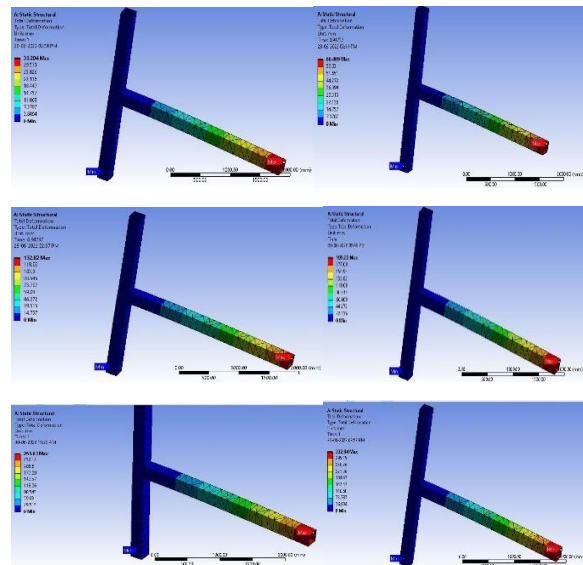


Figure No. 3.1 Channel -1 Total Deformation generated after applied force

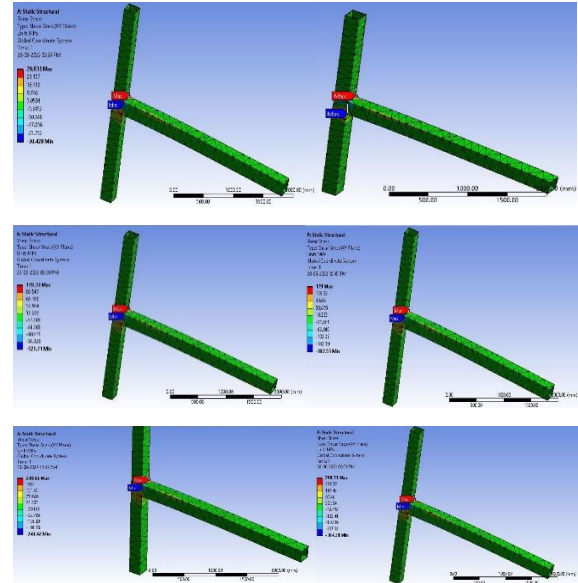


Figure No. 3.2 Channel -1 Shear stress developed due to applied load

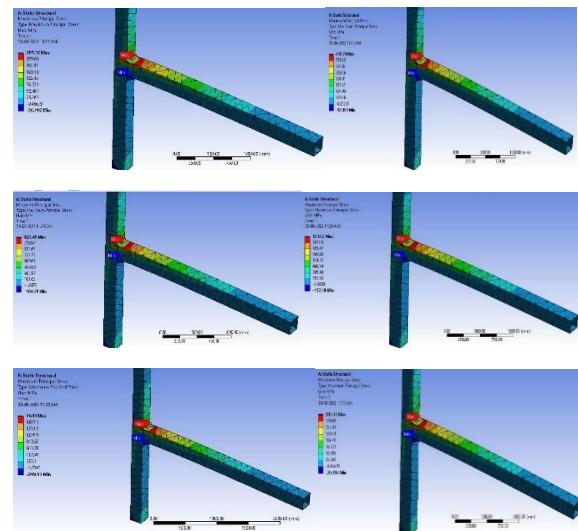


Figure No. 3.3 Channel -1 Maximum Principal stress developed in below section

This method of analysis needed to performed as channel-1 for all the proposed design of structures.

Maximum Shear Stress developed

The data in Table 3.1 clearly demonstrates that, with regard to the ultimate load and deformation at collapse, the joint's performance with the connection details of Channel - 2 outperforms all others. The location of the plastic hinge creation during the inelastic deformation. This information supports

channel - 2 effectiveness because maximum shear stress developed is 167Mpa which is lowest in the critical area. Comparatively another channel produced high stress which is not safe. Figure no. 3.4 shown graph of maximum shear stress

channel - 4 effectiveness because maximum principal stress developed is 1377 Mpa which is lowest in the critical area. Comparatively another channel produced high stress it is not safe. Figure no. 3.5 shown graph of maximum principal stresses.

Table No. 3.1 Maximum Shear Stress developed

Design	Applied Force	Channel - 1	Channel - 2	Channel - 3	Channel - 4
Maximum Shear Stress (Mpa)	50	29	16	46	25
	100	59	33	92	49
	200	119	67	184	101
	300	179	100	276	151
	400	238	134	368	202
	500	298	167	460	253

Tale No. 3.2 Maximum Principal stress

Design	Applied Force	Channel - 1	Channel - 2	Channel - 3	Channel - 4
Maximum Principal stress	50	205	193	212	137
	100	410	387	424	275
	200	821	774	849	550
	300	1232	1161	1275	826
	400	1643	1548	1780	1102
	500	2053	1935	2124	1377

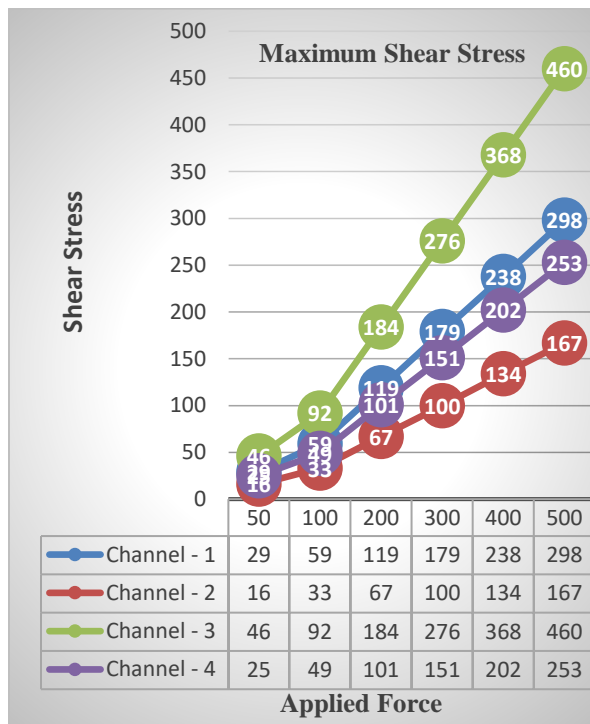


Figure No. 3.4 Maximum shear stress

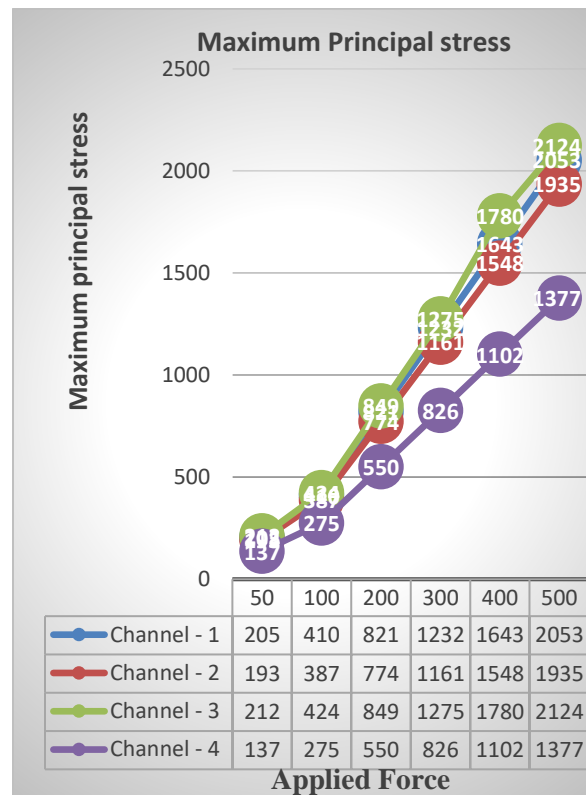


Figure No. 3.5 Maximum Principal stress

Maximum Principal stress developed

The data in Table 3.2 clearly demonstrates that, with regard to the ultimate load and deformation at collapse, the joint's performance with the connection details of Channel - 4 outperforms all others. The location of the plastic hinge creation during the inelastic deformation. This information supports

Total deformation produced

The data in Table 3.3 clearly demonstrates that, with regard to the ultimate load and deformation at collapse, the joint's performance with the connection

details of Channel - 4 outperforms all others. The location of the plastic hinge creation during the inelastic deformation. This information supports channel - 4 effectiveness because total deformation developed is 249mm which is lowest in the critical area. Comparatively another channel produced high deformation it is not safe. Figure No. 3.6 shown total deformation.

Table No 3.3 Total Deformation

Design	Applied Force	Channel - 1	Channel - 2	Channel - 3	Channel - 4
Total Deformation (mm)	50	33	31	28	24
	100	66	65	57	49
	200	132	124	114	99
	300	199	186	171	149
	400	265	248	228	199
	500	298	310	285	249

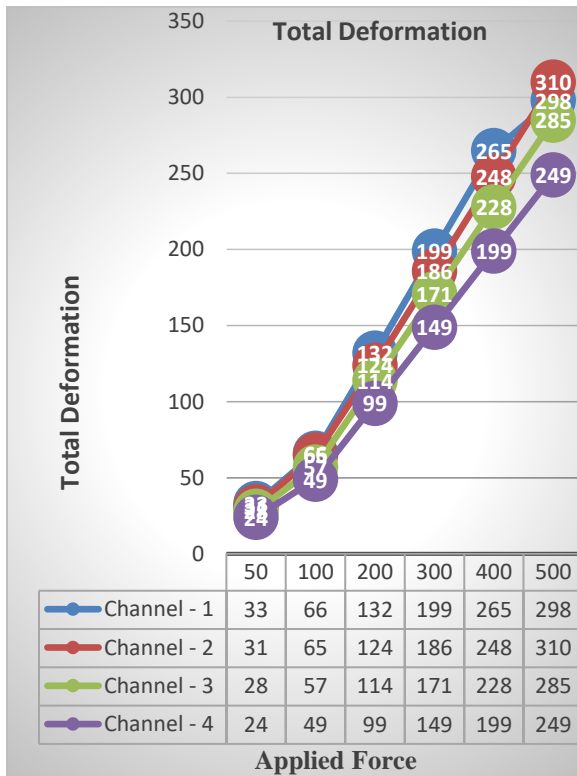


Figure No. 3.6 Total deformation

In this paper total deformation developed in the channel-4 T-shape section is produced comparatively less deformation with respect to another structure and it indicate significantly good result for making square hollow section vertical to horizontally connected beam. It is also observed that channel-4 is capable to bear seismic load.

VII. CONCLUSION

Due to a number of benefits, new hollow steel sections (such as square and rectangular hollow sections) are becoming more common in current steel buildings in India (such as: high strength to weight ratio, higher efficiencies in resisting forces, better fire resistance properties, Higher radius of gyration, lesser surface area). These hollow steel sections do not have standard connection specifications that are listed in design codes or published literature, unlike normal steel sections. To solve this issue, the goal of the current study was to create a practical and affordable connecting detail between two hollow portions that would be capable of smoothly transferring forces and be simple to construct.

A square hollow beam to square hollow column connection was chosen and modelled in the commercial finite element analysis in ANSYS to meet the aforementioned goal. In light of various connection characteristics, this model was examined for nonlinear static (pushover) analysis. For this investigation, the following four alternative connection scheme details were chosen: (i) channel-1 basic structure, (ii) Channel-2 using end plate sections, (iii) channel -3 using closed section and (iv) channel-4 using T-shape plates. The basic design (rectangular hollow beam welded to one face of the rectangular hollow column).

For joints between rectangular hollow beams and rectangular hollow columns, the performance of the chosen connection details is compared, and the best performing connection details are advised.

The following conclusions were reached from studies conducted -

- ❖ The type of connection employed has a significant impact on the joint's maximum deformation capacity and load carrying capability.
- ❖ The joint's maximum allowable shear force ranged from 298 Mpa in the channel-1 basic

model to 253Mpa in channel-4 T-shape section, it is in decreasing order. Similar to this, deformation at collapse ranges from 298 mm in the channel-1 basic model to 199 mm in channel-4 T-shape section is decreased up to safe zone.

- ❖ With the exception of channel-4 T-shape section, all of the connection detail channel show that the creation of the plastic hinge typically takes place at the joint between the beam and column. The plastic hinge in the beam end away from the junction is the outcome of channel-4.
- ❖ Among all joints, channel-4's joint performance (columns jacketed with two channel sections and joined to the beam by welding) exhibits the best performance.

REFERENCE

- [1] Ting LC, Shanmugam NE, and Lee SL., "Box-column to I-beam connections with external stiffeners", *Journal of Construction Steel Research*, Vol. 18.3. pp. 209-226, 1991.
- [2] White RN, Fang PJ, "Framing connections for square structural tubing". *Journal of Structural Engineering*. ASCE, vol. 92, pp. 175-194, 1966.
- [3] Wheeler AT, Clarke MJ, Hancock GJ, Murray TM, "Design model for bolted moment end plate connections joining rectangular hollow sections." *Journal of Structural Engineering*. Vol. 124, pp. 164-173, 1998.
- [4] Chen S, Yeh CH, Chu JM, Chou ZL., "Dynamic behavior of steel frames with beam flanges shaved around connection", *Journal of Construction Steel Research*. Vol. 42, pp. 49-70, 1997.
- [5] Kim, Y.-J., and Oh, S.-H. "Effect of the moment transfer efficiency of a beam web on deformation capacity at box column-to-h beam connections". *Journal of Construction Steel Research*, vol. 63(1), pp.24-36,2007
- [6] P. Ramanathan, I. Baskar, P. Muthupriya, and R.Venkatasubramani, "Performance of SelfCompacting Concrete Containing Different Mineral Admixtures" *KSCE Journal of Civil Engineering* (2013)17(2):465-472.May30 2013.
- [7] Narmluk, Mongkhon and Nawa, Effect of curing Temperature on Pozzolanic Reaction of Fly Ash in Blended Cement Past, *International Journal of Chemical Engineering and Applications*, Vol. 5, pp. 31-35, 2014.

- [8] Singh S.P., and sharan A., Strenght characteristics of compacted pond ash, *Geomechanics and Geoengineering*, 9.1,2014, pp. 9-17.