IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

NUMERICAL INVESTIGATION ON THE FLEXURAL BEHAVIOUR OF COLD FORMED STEEL SIGMA BEAM

[1]K.Subramaniyan, [2]Dr.R.Thenmozhi

[1]PG Scholar, [2]Associate Professor

[1]Structural Engineering,

[1]Government College of Technology Coimbatore, Tamilnadu, India

Abstract: Cold-formed steel sections are widely used for light-gauge structural beams and roof purlins due to their high strength-to-weight ratio and ease of installation on site. The commonly used profiles have a wide variety of cross sectional shapes, e.g. C, Z, 'top hat' and sigma sections. Amongst these popular sections, the sigma section possesses several structural advantages, such as high cross-sectional resistance and large torsional rigidity compared with standard Z or C sections. These also have shear Centre close to the web. Commercially in market, 'Σ' sigma sections are available which are predominantly used as lightly loaded and medium span elements in roofing systems. This study aims to make the flexural member with two point loading, equally strong under compression and tension by opting suitable geometry as Cold Formed Steel has the versatility of being cast into three different sigma sections of equal cross section. The coupon test was conducted to obtain material property. Finite element analysis has been carried out using the software ABAQUS 6.14-1.The investigation has thus enabled to study the effect of dimension and position of web stiffener on the flexural behaviour and strength prediction of cold formed steel sigma section.

***Index Terms* - Beam, Cold-formed steel, Sigma sections, Finite element and Tension coupon test.**

I. INTRODUCTION

In steel construction, there are primarily two types of structural member's hot-rolled steel shapes and cold-formed steel shapes. Hot-rolled steel shapes are formed at elevated temperatures while cold-formed steel shapes are formed at room temperature, thus the name cold-formed steel. Cold-Formed Steel (CFS) structural members, less familiar but of growing importance. Cold-Formed Steel members are commonly used as purlins, cladding rails, sheeting rails, wall studs, floor joists, sheets and decks etc. When compared with thicker hot rolled members they provide a substantial increase in strength to weight ratio. In market various shapes of these products are available and 'C' and 'Z' sections are predominantly used in light load and medium span situation such as roof systems.

The use of cold-formed steel members in building construction began in the 1850s in both the United States and Great Britain. In the 1920s and 1930s, acceptance of cold-formed steel as a construction material was still limited because there was no adequate design standard and limited information on material use in building codes. One of the first documented uses of cold-formed steel as a building material is the Virginal Baptist hospital, constructed around 1925 in Lynchburg, Virginia. The walls were load bearing masonry, but the floor system was framed with double back-to-back cold-formed steel lipped channels, Greene engineered a recent renovation to the structure and said that for the most part, the joists are still performing well. Today it finds many applications in different types of industries, various types of equipment's, construction of car and motor vehicle bodies, railway coaches, storage racks, highway products and bridge construction.

The method of cold forming points at the process of metal forming in which the shape and structure of steel is being changed by drawing, extruding, hammering, pressing, spinning or stretching at temperatures below the steel's recrystallization temperature. These actions create alterations in the metal's work which increase its hardness and tensile strength while enhancing the surface finish. These are given the generic title Cold Formed Steel sections. Sometimes they are also called Light Gauge Steel sections or Cold Rolled Steel sections. The thickness of steel sheet used in Cold Formed construction is usually 1 to 3 mm. The method of manufacturing is important as it differentiates these products from Hot Rolled Steel sections.

II. MATERIAL PROPERTY

The material of the beam adopted as steel. The modulus of elasticity kept as 210 GPa and poisson's ratio as 0.3. In ABAQUS, the stress strain data has to be entered as true stress and true plastic strain. Since the investigation is aimed to achieve efficient and economic section in relative to other sections, the stress strain data available in the laboratory was taken for the finite element analysis. Following expressions were used to convert the nominal stress to true stress, nominal strain to true strain and true strain to plastic strain.

$$\sigma_T = P/A = \frac{P}{Ao} x \frac{l}{lo} = \sigma_E\,(1 + \varepsilon_E\,)$$

$$\varepsilon_E = ln\frac{lf}{lo} = ln\frac{l+\Delta l}{lo} = \ln\,(1+ \varepsilon_E\,)$$

$\varepsilon_{PT} = \varepsilon_T - (\sigma_T/E_S)$

Table 1: Coupon test results

S. No	Length (mm)	Width (mm)	Thickness (mm)	Young's Modulus 'E' (MPa)	Fy (MPa)	Yielding Type
1	125	25	2	2.04×10^5	240	G
2	125	25	2	2.06×10^5	240	G
3	125	25	2	2.05×10^5	250	G

III. NUMERICAL INVESTIGATION

3.1 Introduction of FEM

The Finite Element Method (FEM) is a numerical analysis for obtaining approximate solutions to a wide variety of engineering problems. The finite element method is now accepted as the most powerful general technique for the numerical solution of a variety engineering problems. Applications range from the stress analysis of solids to the solution of acoustical phenomena, neutron physics, heat transfer problems etc.

The FEM (Its practical application often known as Finite Element Analysis) is a numerical technique for finding approximate solutions of partial differential equations (PDE) as well as integral equations. In FEA, a complicated structure is divided into smaller elements. Each element is based on physical laws using numerical computing techniques. All the elements are assembled into a big matrix of algebraic equations. This matrix is usually solved by computer. Finally, the solution is obtained according to the engineer's requirements.

Advances, in the field of computer aided engineering during the last two decades have been quite extensive and have led to considerable benefits to many engineering industries. In the building industry, use of advanced finite element tools has not only allowed the introduction of innovative and efficient building products, but also the development of accurate design methods. High-performance computing facilities and advanced finite element programs are now available for research and development activities in many universities in collaboration with industries. Cold-formed steel building system was accurately simulated by finite element models and thus considerably reduced the number of time consuming and expensive large scale experiments required.

3.2 Abaqus Programme

The finite element program ABAQUS is a computational tool for modeling structures with material and geometric nonlinear behavior. ABAQUS version 6.14-1 was used extensively to simulate the behavior and strength of cold-formed steel beams under flexural loading conditions ABAQUS is a commercially available finite element analysis software package for FEA. It is a general purpose Finite Element Modeling package for numerically solving a variety of mechanical problems. These problems include static and dynamic structural analysis (both linear and non-linear), steady state and transient problems, mode frequency and buckling analysis, acoustic and electromagnetic problems and various types of field and coupled-field applications. The program contains many special features which allow nonlinearities or secondary effects to be included in the solution such as plasticity, large strain, hyper elasticity, creep, swelling, large deflections, material anisotropy and radiation. The finite element program of ABAQUS 6.14-1 versions used to develop the finite element model, which aimed to simulate the behavior of the cold formed steel sigma flexural members. A graphical user interface (GUI) is available throughout the program, to guide new users through the learning process and provide more experienced users with multiple windows, pull-down menus, dialogue boxes, tool bars and on-line documentation.

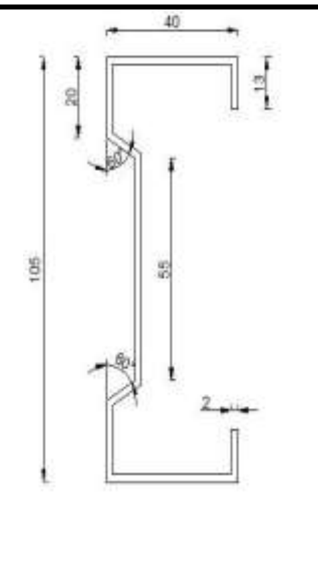
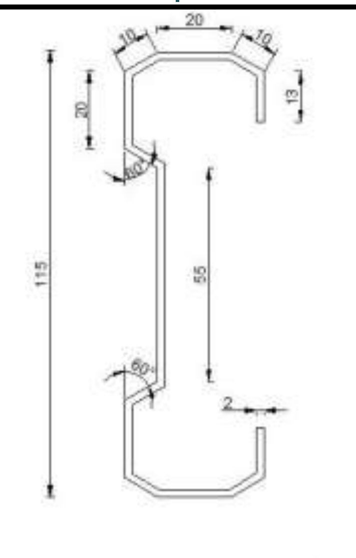
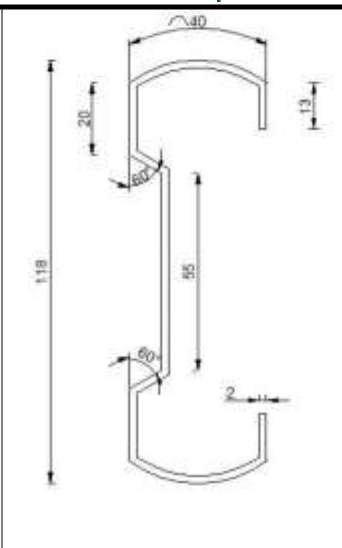
3.3 Development of the Sigma section Beam

3.3.1 Section Geometry

In this investigation In the 3 different sigma section profiles of equal cross sectional area having span of 2100 mm have been modeled using finite element software ABAQUS 6.14 with thickness as 2mm and keeping the weight of the beam as constant. Modeling is done in ABAQUS 6.14 with 3D- Deformable SHELL ELEMENT. Table 2 shows the Geometry of the proposed sigma section.

Table 2: Details about beam

S. No	1	2	3
Beam id	S1	S2	S3
Type of flange	Flat flange	Pentagon flange	Curved flange
h x b x d (mm)	105 x 40 x 20	115 x 37.32 x 20	118 x 37.32 x 20
t (mm)	2	2	2
l (mm)	2100	2100	2100

Profile	 FLAT FLANGE (S1)	 PENTAGON FLANGE (S2)	 CURVED FLANGE (S3)

3.4 Development of the Finite Element Model

3.4.1 Modeling & Meshing

The modeling of sigma sections beam was started by creating three dimensional, deformable SHELL part in ABAQUS. The shell element was used in all the finite element models. Finite element models simulating the simply supported boundary conditions and center point loading were developed. The size of the mesh taken as 10 mm. surface-to-surface interactions was created between the stiffener and beam section & tie constrains are given at both ends of the section.

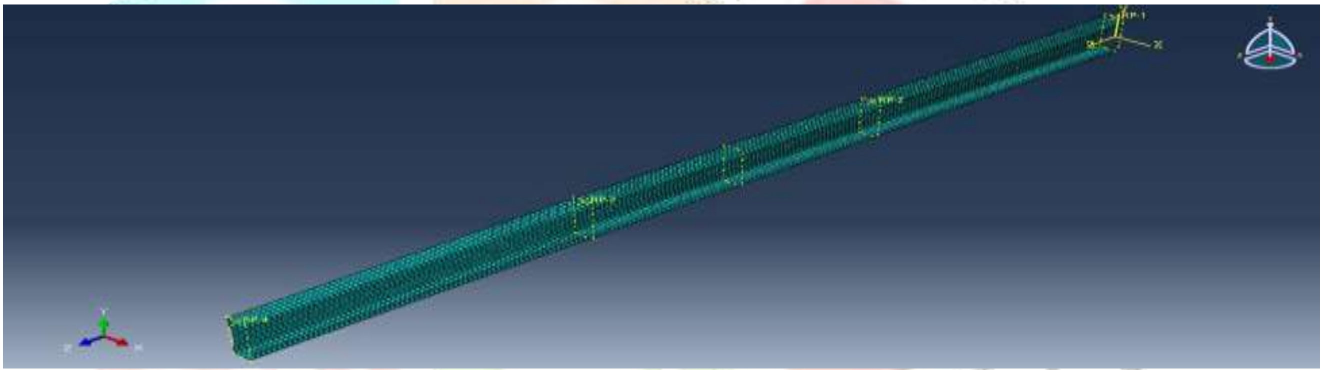


Figure 1: Modeling and meshing of specimen S1

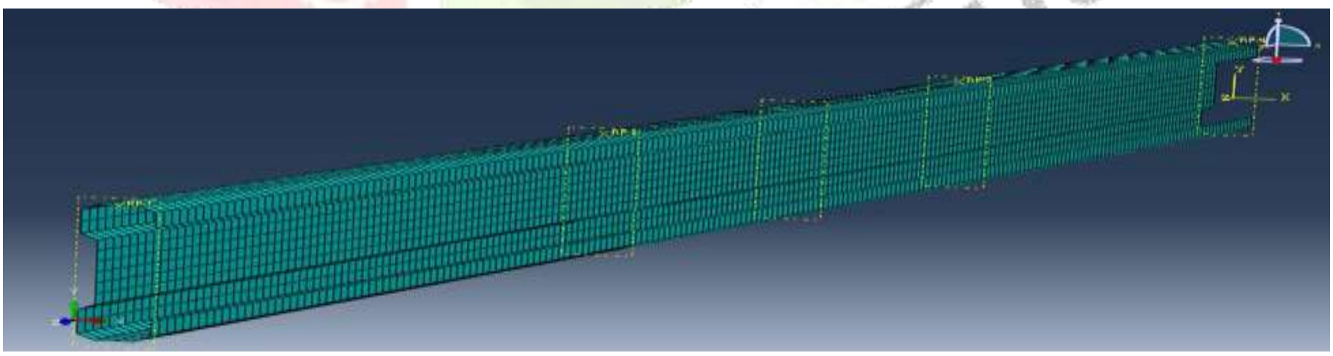


Figure 2: Modeling and meshing of specimen S2

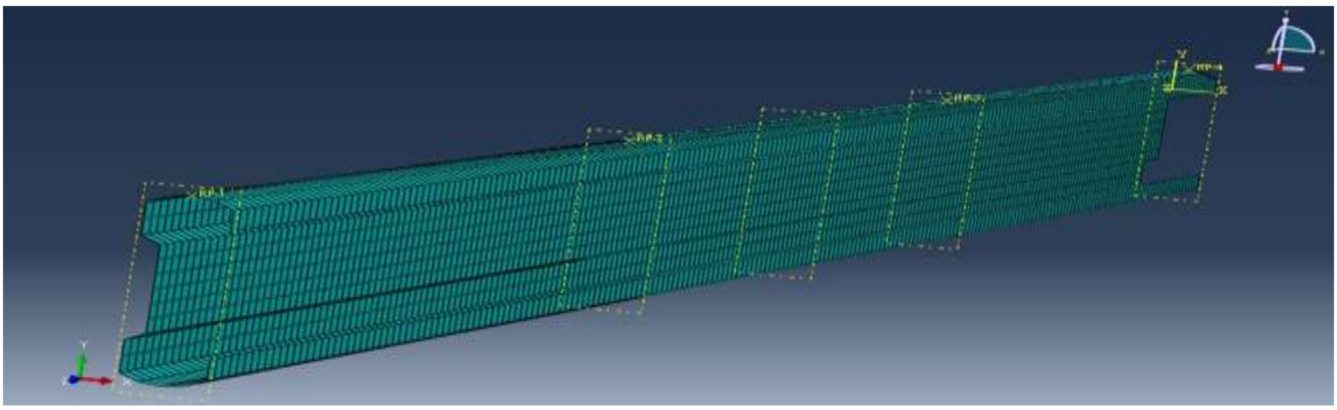


Figure 3: Modeling and meshing of specimen S3

3.4.2 BOUNDARY CONDITIONS

Boundary conditions were assigned to each member of the instance

- One end of the beam section was hinged at the base (U1=0; U2=0; U3=0)
- Other end of the beam section was given roller support at the base (U1=0; U2=0; U3=0).

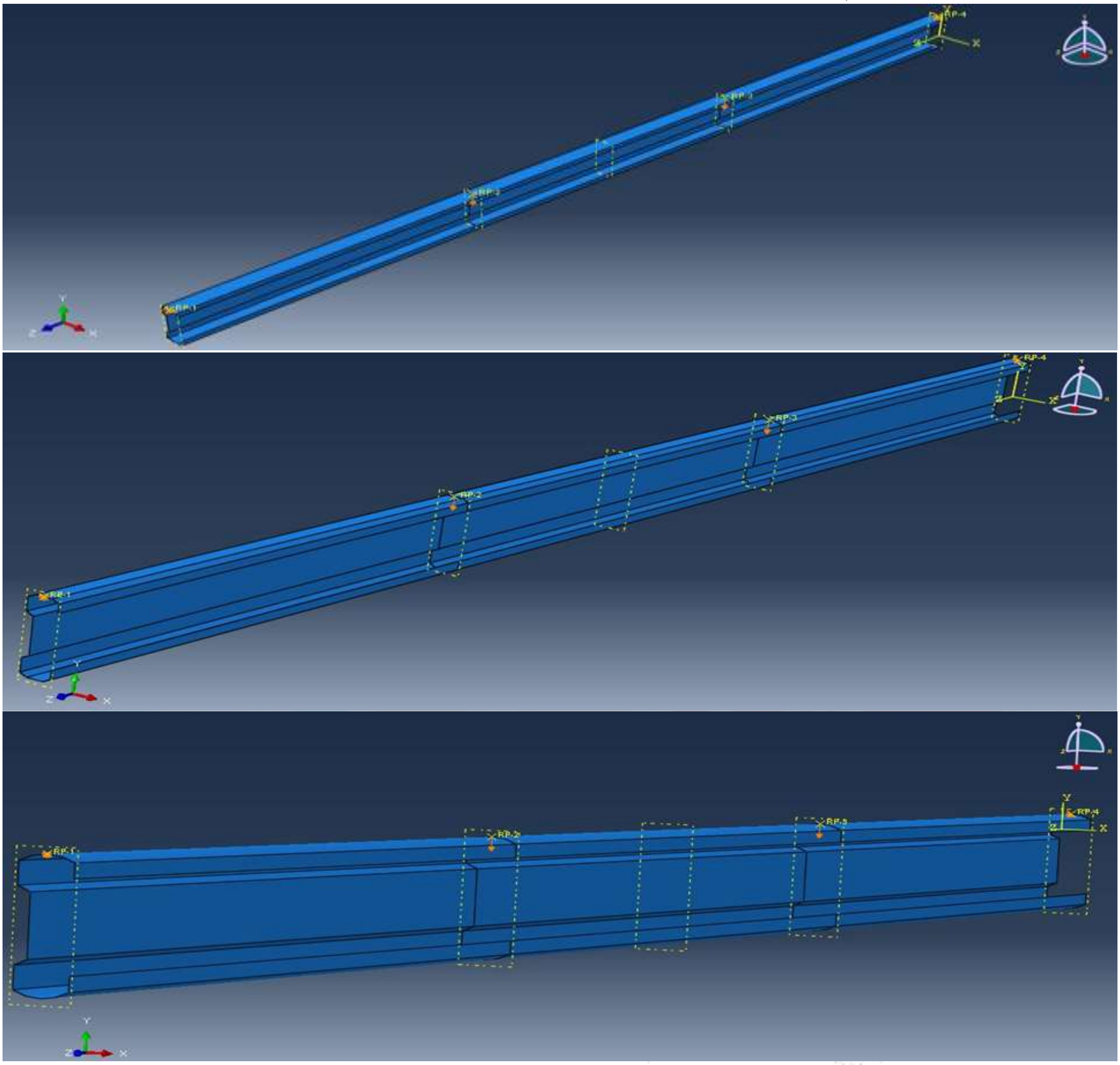
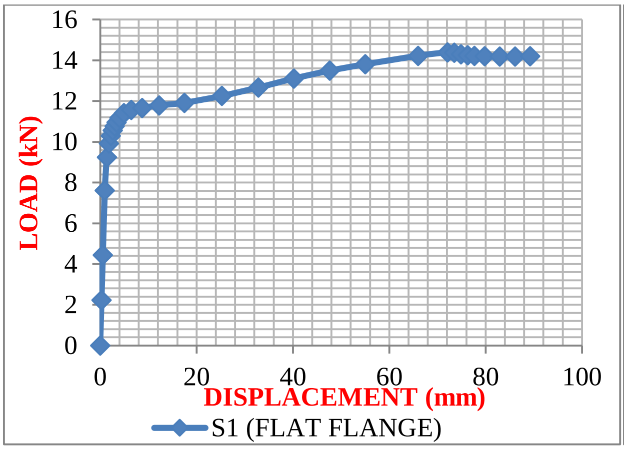
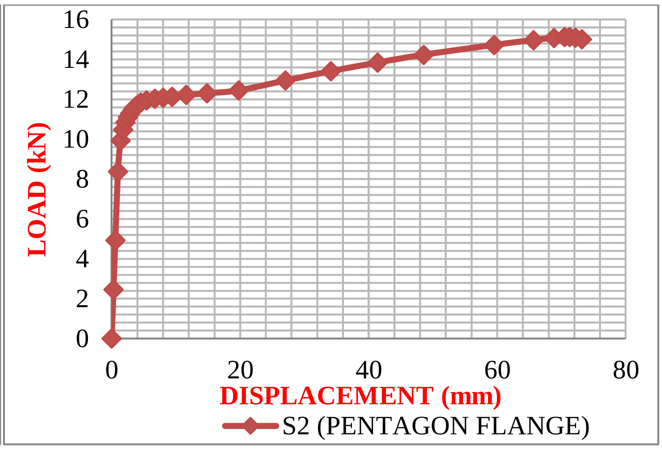


Figure 4: Boundary conditions of specimens

IV. RESULTS AND DISCUSSION

Finite element software ABAQUS 6.14-1 has been employed in determining the moment carrying capacity for the cold-formed sigma section. From this analysis, vertical deformation, Von mises stresses were obtained. They were discussed below. From the chart, the difference in percentage values for deflection has been derived.





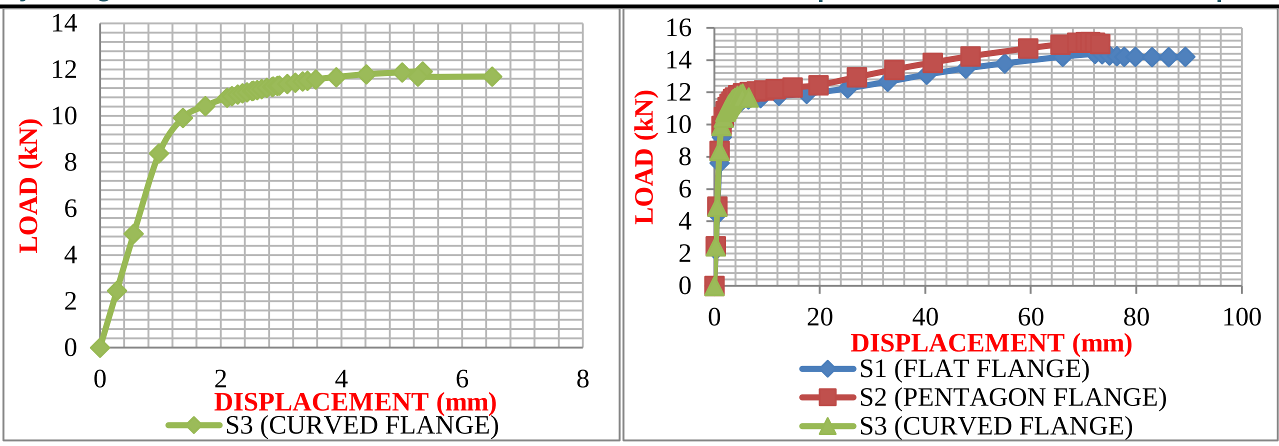


Figure 5: Load vs Displacement curve

The results of load and displacement component for all the three sigma sections are plotted in the above figure 5. It is observed that the specimen S1 contain up to 9.2kN material is elastic and beyond its limit plastic manner, then specimen S2 contain up to 9.6kN material is elastic and beyond its limit plastic manner and specimen S3 contain up to 8.4kN material is elastic and beyond its limit plastic manner. It is observed that the specimen S2 (Pentagon flange) carrying maximum load.

4.1 Von mises stress contour

The following figure shows the contour plot of Von-Mises stress for various sigma sections. The Von-Mises stress is maximum at the top portion of flange at mid span.

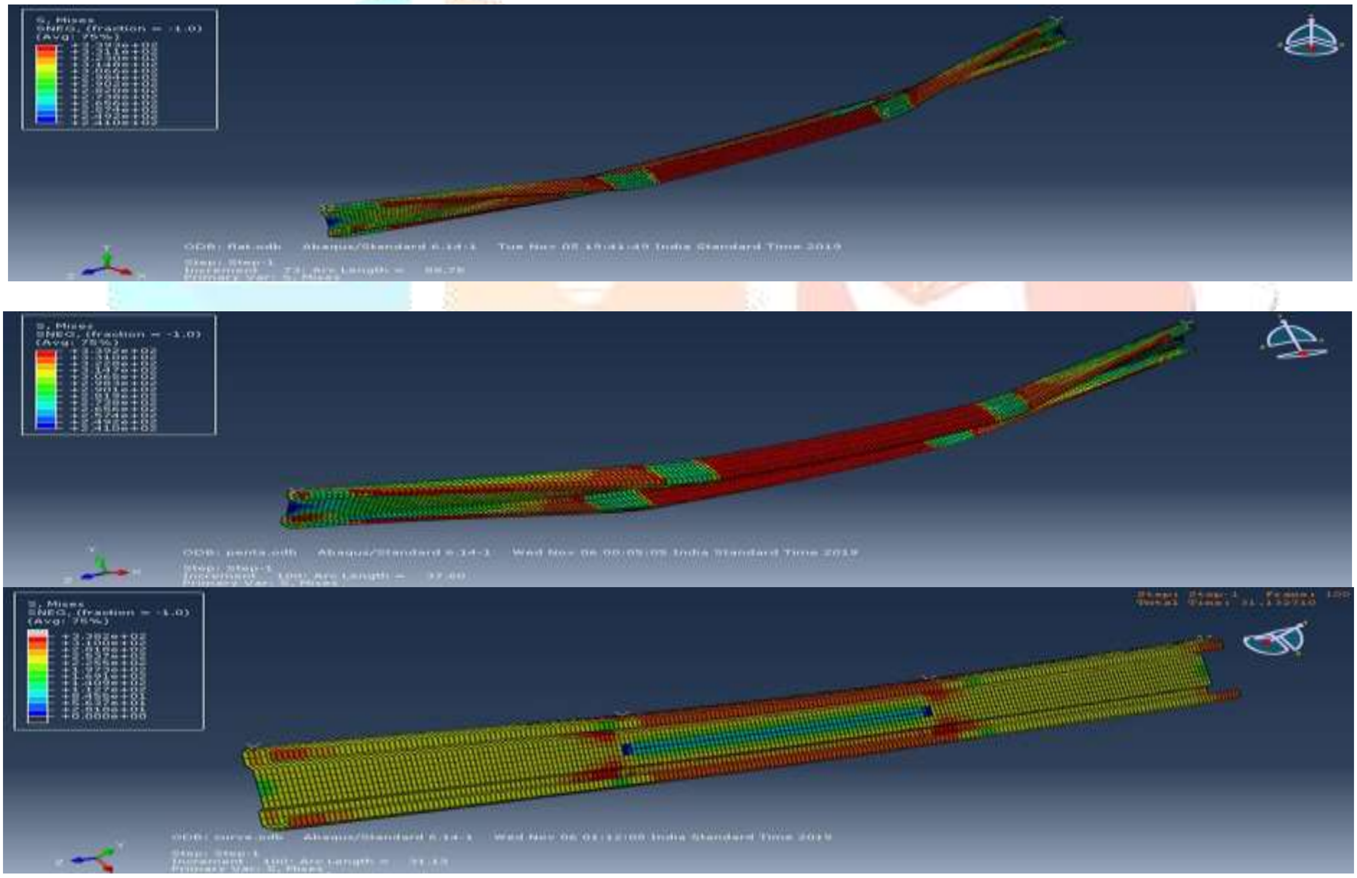


Figure 6: Von mises stress contour for specimen S1, S2 & S3

From the above Figure, It can be seen that the specimen S2 attains the maximum load carrying capacity. The specimen 3 undergoes lateral torsional buckling behaviour and a specimen 1 and 2 undergoes local buckling behavior at near load points.

V. SUMMARY AND CONCLUSION

- In this study it is aimed to investigate the flexural behaviour of cold-formed steel sigma sections.
- Finite element analysis is carried out for three different specimens using ABAQUS 6.14.

The following conclusions are drawn from the numerical investigation.

- Numerical investigation showed that specimen S2 (Pentagon flange) having high load carrying capacity compared to other sections.
- It is observed that the specimen S3 (curved flange) behaves stiffer manner compared to other sections.
- When comparing the results, the specimen S3 (Curved flange) undergoes lateral torsional buckling behaviour and specimen S1 (Flat flange) and S2 (Pentagon flange) undergoes local buckling behavior at near load points.

REFERENCES

[1] Anna Green Antony (2016), "Study on Cold Formed Steel Sigma Sections and the Effect of Stiffeners", Vol.5, Issue 9.

[2] Ben Young and Gregory J. Hancock (2000), "web crippling behavior of channels with flanges restrained", Journal for Constructional Steel Research.

[3] DinisPB, Camotim D. Local/distortional mode interaction in cold-formed steel lipped channel beams. Thin Walled Structures 2010; 48:771-85.

[4] G.J. Hancock (2003), "Cold-formed steel structures" (Journals of Constructional Steel Research).

[5] IS 801-1975, "Specification for Cold-formed light gauge structural steel sections".

[6] IS 811-1987, "Specification for Cold-formed light gauge structural steel sections".

[7] Jian Yang, Qiang Liu (2012), "Sleeve connections of cold-formed steel sigma Purlins Engineering Structures", 43 (2012), 245-258.

[8] Jian Yang, Qiang Liu (2012), "An experimental study into flexural behavior of sigma purlins attached with roof sheets Engineering Structures", 45 (2012) 481-495.

[9] Katarzyna ciesielczyk, katarzyna rzeszut (2016), "Local and distortional buckling of axially loaded cold rolled sigma profiles".

[10] Long-Yuan Li (2009), "Analysis of distortional buckling of cold-formed sigma purlins using EN1993-1-3", Journal of Constructional Steel Research (2009) 2099-2102.

[11] V. Raghul, N. Uma Maheswari (2015), "Analytical Investigation on Cold-Formed Steel Built-Up Section Under Flexure Vol. 3 Issue 4".

[12] Schafer BW. Cold-formed steel behavior and design: Analytical and Numerical Modeling of Elements and Members with Longitudinal Stiffeners. Cornell University. Ithaca, New York, USA; 1997.

[13] Schafer BW. CUFSM: Elastic Buckling Analysis of Thin-Walled Members by finite strip analysis. CUFSM. Version 4.04, Mar 2012.