Comparison study of Experimental and ANSYS Results of Cold Formed Steel Angle Sections

Paul Makesh A[1], Arivalagan S[2]
[1]Research Scholar, Civil Engineering, Dr.M.G.R Educational and Research Institute University, Maduravoyal, Chennai, India
[2]Dean and Professor, Civil Engineering, Dr.M.G.R Educational and Research Institute University, Maduravoyal Chennai, India

Abstract- **The effective sectional area concept was adopted to conduct the analysis of cold-formed Tension members. ANSYS software was utilized to simulate the behavior of cold formed steel angle under tension load. The paper describes the results from a finite element investigation into the load capacity tension members of single angle sections of 2mm and 4mm and double angles sections of 2mm and 4mm under plain (without Lipped) and with Lipped conditions subjected to tension. Results were recorded as the load carrying capacity increases for connected to the opposite side of the gusset than the connected to same side. Comparison between Experimental Load, ANSYS Load. Results of Finite Element Analyses are compared with experimental results.**

Index Terms- ANSYS, Tension members; Cold-formed steel, strength, displacement.

I. INTRODUCTION

Cold formed steel member are less weight and thinner than hot- rolled sections. They can be used to produce and forming of almost any shapr and section to any desired geometry and length. Openings of cold formed steel beams used to facilitate sanitary, electrical and mechanical works. These openings should have size, shape and location, as far as possible; have no effect on the structural strength requirements. The main disadvantages of opening in cold formed steel sections are the local buckling due to high width of open to thickness ratios. Recent codes of practice and standards have suggested simplified methods and processes for the design of steel members with perforation. However, numerical and experimental researches have been published to investigate the effect of openings on the load capacity of cold formed steel (CFS) members subjected to monitonic axial load. An extensive parametric study have helped to enhance the understanding the behaviour and buckling of wide range of opening angle sections under different combinations of axial tension load moment. Numercial modeling is one of the important features in finite element analysis. This chapter discusses the finite element modeling of the cold formed steel angles, the finite element analysis program ANSYS is used to create the model of the tested specimens Under these models, ultimate loads and total deformation of cold formed angles are compared with experimental results angles.

ANSYS Workbench capabilities include a unique and extensive materials and sections for concrete and steel structures.. A user- friendly beam and shell postprocessor included listing and plotting section geometry, reinforcements, beam stresses and strains inside the cross section. The skilled combination module, selects loads and coefficients for logic code combinations. Results embrace concomitance. The analysis is carried out in three stages such as. 1. Preprocessor 2. Solution 3. Post processor.

II. LITERATURE REVIEW

In order to understand flexural behavior of CFS members and why there is need of this study, a through literature review was undertaken. This literature review included review of the characteristics, design methods and numerical methods to analyze and accurate modeling of CFS sections followed by a summary which presents main findings and gaps in the literature.

Alireza Bagheri et.al [1] (2012) are presented the Cyclic behavior of bolted cold – formed angles. In this paper a finite element (FE) procedure is described for simulating hysteretic moment – rotation behavior and failure deformations of bolted cold-formed steel (CFS).

K.F. Chung and K.H,Ip [2] (2012) are presented the Finite element investigation on the structural behavior of cold formed steel bolted connections. A finite element model with three-dimensional soild elements established to investigate the bearing failure of cold- formed steel bolted connections.

Valdier Francisco et al.[3] presented details of 66 experiments carried out on cold formed steel fastened with bolts subjected to tension. They examined the reduction coefficient performance based upon the new tests and data available in the literature, comprising of 108 tests.

Chi-Ling Pan[4] investigated the effect of shear lag on the angles cold formed steel sections, by testing 54 specimens with different cross sectional dimensions. The Indian code for use of cold formed steel IS: 801[5] does not any provision for the design of tension members. Hence during the code revision, experiments were conducted at CSIR-SERC on cold formed angle tension members for the inclusion of design provisions.

III.EXPERIMENTAL INVESTIGATION

A total of 72 specimens have been tested by varying the angle sizes, number of bolts and the bold pitch distance. All the specimens have been designed to undergo net section rupture failure or block failure. The specimens are equal angles 50x50, 60x60 and 70x70mm, and unequal angles are 50x25,60x30 and 70x35mm they have equal length and thickness of 500mm and 2mm respectively. The angles are connected to the gusset plate under eccentric tensile loads on single and double angle specimen. The ress vs strain curve was plotted as shown in Fig 1.

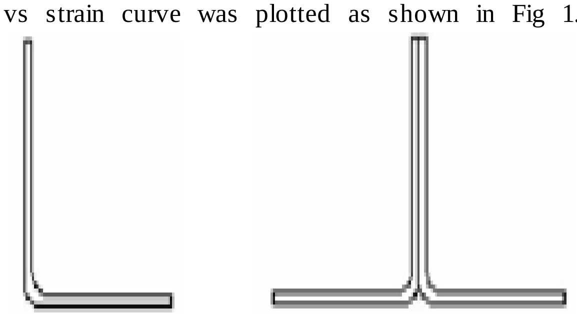

Fig 1 Single angles and Double angles

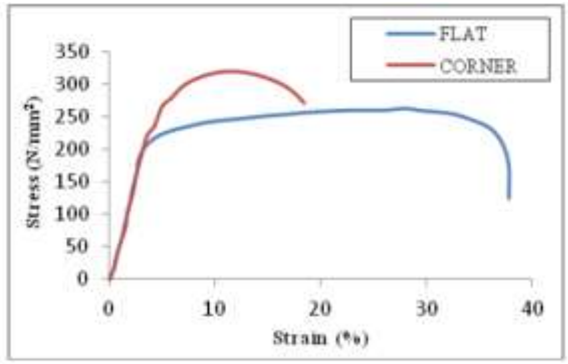

Fig 2 Stress vs Strain

IV.FINITE ELEMENT ANALYSIS

The goal of the Finite Element Analysis was to develop a model that could study the effect of connection eccentricity and connection length on tension member. In order to test the validity, the results obtained from the analysis were used to compare with the test results from the experiments. The Finite Element Analysis was performed using the commercial Finite Element Program ANSYS, version 16.0.

FEM is best understood from its practical application, known as finite element analysis (FEA). FEA as applied in engineering is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm. In applying FEA, the complex problem is usually a physical system with the underlying physics such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed in either PDE or integral equations, while the divided small elements of the complex problem represent different areas in the physical system.

FEA is a good choice for analyzing problems over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. FEA simulations provide a valuable resource as they remove multiple instances of creation and testing of hard prototypes for various high fidelity situations

V. APPLICATION

A variety of specializations under the umbrella of the mechanical engineering discipline (such as aeronautical, biomechanical, and automotive industries) commonly use integrated FEM in design and development of their products. Several modern FEM packages include specific components such as thermal, electromagnetic, fluid, and structural working environments. In a structural simulation, FEM helps tremendously in producing stiffness and strength visualizations and also in minimizing weight, materials, and costs.

FEM allows detailed visualization of where structures bend or twist, and indicates the distribution of stresses and displacements. FEM software provides a wide range of simulation options for controlling the complexity of both modeling and analysis of a system. Similarly, the desired level of accuracy required and associated computational time requirements can be managed simultaneously to address most engineering applications. FEM allows entire designs to be constructed, refined, and optimized before the design is manufactured.

Table 1: Design strength valves in ANSYS

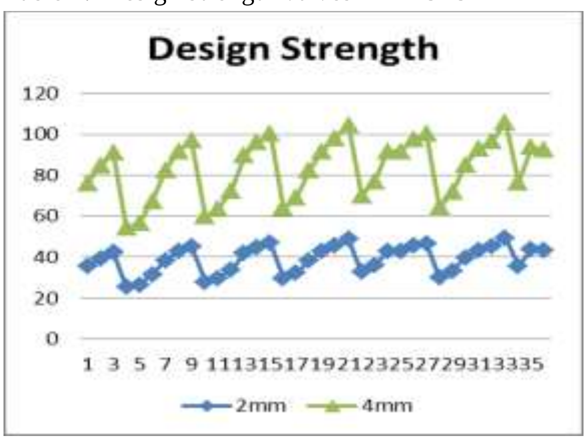

Fig 3 Comparison of Ultimate valves of 2mm and 4mm

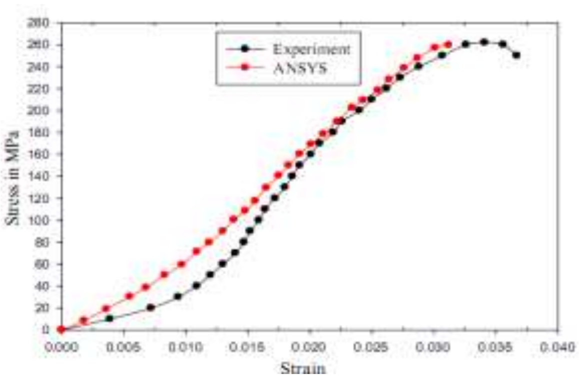

Fig 4 Comparison of Stress and strain

VI NUMERICAL INVESTIGATION

To validate the experimental results, a finite element analysis package ANSYS (16.2) was used for the modeling and analysis. A non-linear analysis was performed and the materials are assumed to behave as an isotropic hardening material. From the experimental tension test results, the static material modeling was done. The element type used to model the test specimens is SHELL 63. It is a 4-noded 3 dimensional quadratic shell element. This element has six degrees of freedom at each node. Finite element mesh of size 2x2mm was implied and used in all the simulations. The friction or contact between connected leg of the specimen and the gusset plate was ignored. Figure 3 shows the single angle without Lip, the load applied on the element

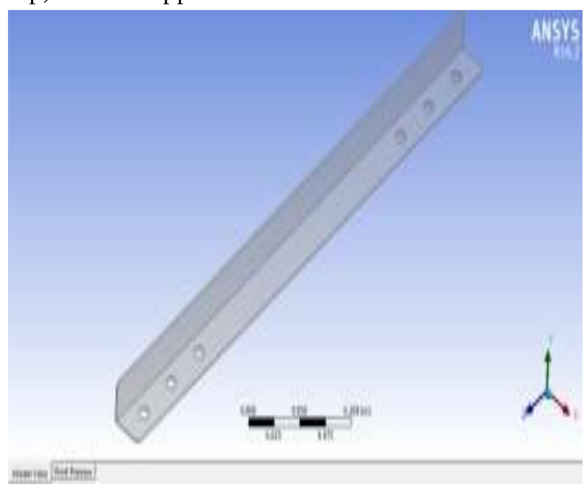

Fig 5 Single angle without Lip 50x50x2 (Ansys)

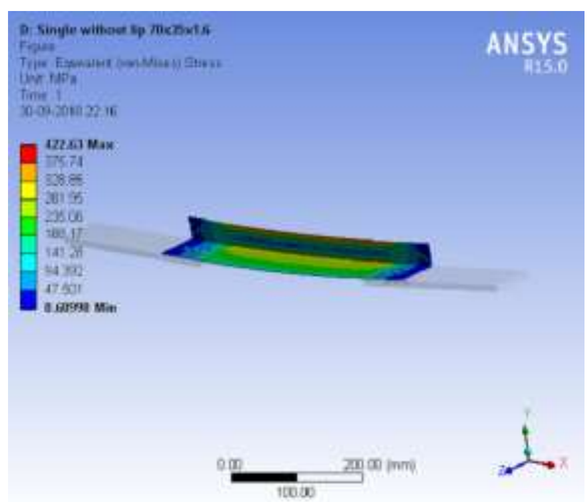

Fig 6 Single angle without Lip 70x35x2 (Ansys)

S.No	Description	Size of Specimen	Ultimate Strength (P_{Ds}) 2mm (KN)	4mm
1	Single angle without Lip	50x50	34.58	72.28
		60x60	38.13	83.57
		70x70	45.26	92.86
		50x25	41.25	90.12
		60x30	51.47	96.45
		70x35	62.47	115.23
2	Single angle with Lip	50x50x10	75.48	162.58
		60x60 x10	87.29	192.58
		70x70 x10	108.1	228.25
		50x25 x10	80.47	164.58
		60x30 x10	92.58	184.52
		70x35 x10	112.2	238.12
3	Double angle on opposite side without Lip	50x50	86.27	198.52
		60x60	92.47	234.56
		70x70	117.2	262.59
		50x25	87.46	197.82
		60x30	98.75	234.58
		70x35	112.1	287.25
4	Double angle on opposite side without Lip	50x50	28.17	55.48
		60x60	32.78	75.28
		70x70	37.85	88.54
		50x25	42.58	76.28
		60x30	48.75	89.56
		70x35	53.78	96.48
5	Double angle on same side with Lip	50x50 x10	59.76	105.48
		60x60 x10	73.70	137.45
		70x70 x10	87.81	175.28
		50x25 x10	58.81	109.48
		60x30 x10	62.78	135.48
		70x35 x10	83.47	172.58
6	Double angle on same side with Lip	50x50 x10	72.37	132.58
		60x60 x10	89.34	158.53
		70x70 x10	106.3	184.31
		50x25 x10	71.58	148.47
		60x30 x10	83.47	175.48
		70x35 x10	92.47	206.48

Table 1 Design Strength of Experimental Results

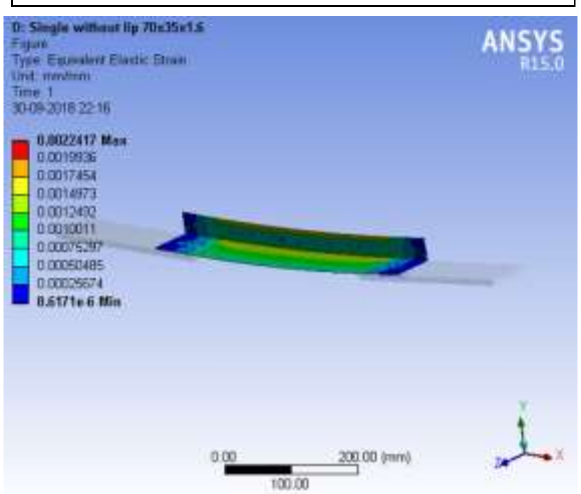

Fig 7 Single angle without Lip 60x30x2

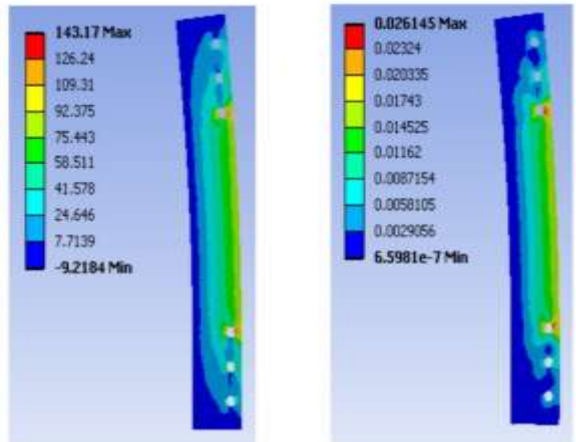

Fig 8 Single angle without Lip, 50 x 50 x4

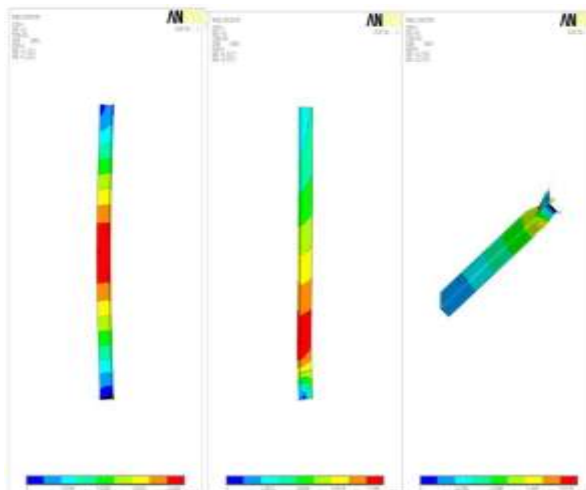

Fig 9 Single angle without Lip 60 x 60 x 4

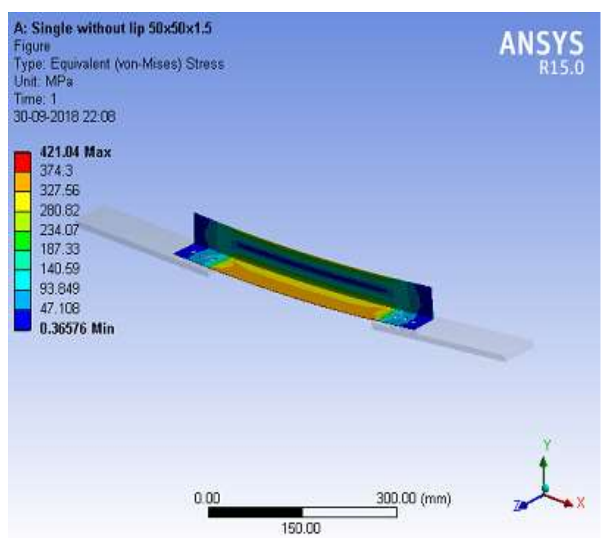

Fig 10 Single angle without Lip 60x30x4

S.No	Description	Size of Specimen	Ultimate Strength (P_{DS})	
			2mm (KN)	4mm
1	Single angle without Lip	50x50	36.78	75.28
		60x60	40.72	87.13
		70x70	47.82	97.29
		50x25	43.72	94.28
		60x30	54.28	100.26
		70x35	65.21	121.29
2	Single angle with Lip	50x50x10	77.82	170.28
		60x60 x10	90.72	200.78
		70x70 x10	113.0	235.46
		50x25 x10	83.78	170.29
		60x30 x10	95.74	190.26
		70x35 x10	115.8	245.29
3	Double angle on opposite side without Lip	50x50	91.26	205.89
		60x60	95.72	245.29
		70x70	120.7	273.29
		50x25	90.78	206.48
		60x30	103.4	243.18
		70x35	115.8	296.18
4	Double angle on opposite side without Lip	50x50	29.46	57.29
		60x60	34.72	79.29
		70x70	39.72	92.58
		50x25	44.88	81.28
		60x30	51.89	94.28
		70x35	56.79	102.25
5	Double angle on same side with Lip	50x50 x10	63.73	112.28
		60x60 x10	77.89	145.28
		70x70 x10	92.18	182.58
		50x25 x10	61.89	115.29
		60x30 x10	65.73	143.18
		70x35 x10	87.18	178.29
6	Double angle on same side with Lip	50x50 x10	76.83	138.48
		60x60 x10	93.73	166.29
		70x70 x10	111.5	190.89
		50x25 x10	75.29	155.29
		60x30 x10	88.79	183.28
		70x35 x10	97.83	214.29

Table 2 Design Strength of Ansys Results

VII CONCLUSIONS

Based on the experimental, numerical and analytical results were concluded. Experimental results shown that the ultimate strength of single equal angle lipped section under tension load is increase 1.26 times greater than single equal plain angle section

1. In the case of single unequal angle lipped section under tension load is increase 1.24 times more than single unequal plain angle section.
2. Numerical results shown that the ultimate strength of single equal plain angle sections without lip 5% higher than the experimental loads under tension.
3. To examine that the single equal angle lipped section under tension load is increase 4% times greater than experimental loads.In the case of Double angles specimens connected to opposite side without Lip 6% higher than the experimental loads.
4. Also it was observed that Double angles specimens connected to same side without Lip 5% higher than the experimental loads. In the case of Double angles specimens connected to opposite side without Lip 5% higher than the experimental loads. Also it was observed that Double angles specimens connected to same side without Lip 6% higher than the experimental loads.
5. The stress contours obtained in the finite element analysis indicates that maximum stresses occur in the innermost bolt holes from which the experimental failures were initiated.

REFERENCES

[1] AS/NZS: 4600 (2005), 'Cold-formed Steel Structures', Australia / New Zealand Standard.

[2] BS: 5950-Part 5 (1998), 'Structural Use of Steelwork in Building-Code of practice for design of cold-formed thin gauge sections', British Standards Institution.

[3] Gupta L.M and Mohan Gupta (2005), "Limit state design of bolted steel angles under tension", Journal of Structural Engineering, Vol.31, No.4, pp265-274.

[4] AISI (1996). Cold-Formed Steel Design Manual, American Iron and Steel Institute, Washington,

[5] AISI (2001). Cold-Formed Steel Standard, American Iron and Steel Institute, Washington,

[6] ANSYS Release 8.1 (2004). ANSYS, Inc., USA.

[7] AS/NZS (1996). Australian/New Zealand Standard, Cold- Formed Steel Structures, Sydney, Australia.