International Journal of Research and Review
www.ijrrjournal.com E-ISSN: 2349-9788; P-ISSN: 2454-2237

Research Paper

Structural Performance of Cold Formed Steel Angle

A Paul Makesh[1], Dr. S Arivalagan[2]

[1]Research Scholar, Department of Civil Engg, [2]Professor & Dean, Head of Department in Civil Engg,
Dr.M.G.R Educational and Research Institute, Maduravoyal, Chennai, India

Corresponding Author: A Paul Makesh

ABSTRACT

The paper describes the results from a finite element investigation into the load capacity tension members of single angle sections of 2, 3 mm 4 and double angles sections of 2mm 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. Results of Finite Element Analyses are compared with experimental results.

Keywords: CFS, ANSYS Workbench, element, mesh

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 requirments. The main disadvantages of opening in cold formed steel sections is 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 investigated 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.

EXPERIMENTAL PROGRAMME

A total of 108 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.

NUMERICAL PROGRAMME

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.

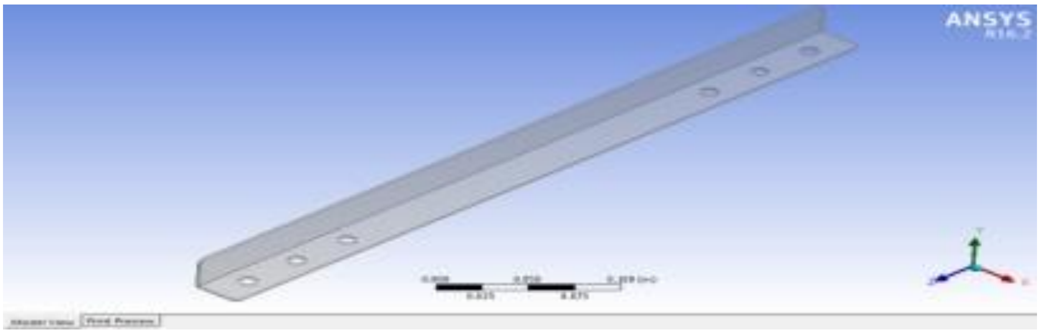

Fig 1 Single angle without Lip 50x50x2 (Ansys)

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.

Table 1: Design strength valves in ANSYS

S.No	Description	Size of Specimen	Design Strength (P_{DS})		
			2mm	3mm	4mm
1	Single angle without Lip	50x50	35.46	37.94	40.60
		60x60	39.45	42.21	45.17
		70x70	42.42	45.39	48.57
		50x25	25.34	27.11	29.01
		60x30	26.48	28.33	30.32
		70x35	31.45	33.65	36.01
2	Single angle with Lip	50x50	38.45	41.14	44.02
		60x60	42.78	45.77	48.98
		70x70	45.26	48.43	51.82
		50x25	27.89	29.84	31.93
		60x30	29.74	31.82	34.05
		70x35	33.78	36.14	38.67
3	Double angle on opposite side without Lip	50x50	41.78	44.70	47.83
		60x60	44.78	47.91	51.27
		70x70	46.85	50.13	53.64
		50x25	29.78	31.86	34.10
		60x30	32.14	34.39	36.80
		70x35	38.42	41.11	43.99
4	Double angle on opposite side with Lip	50x50	42.74	45.73	48.93
		60x60	45.78	48.98	52.41
		70x70	48.78	52.19	55.85
		50x25	32.74	35.03	37.48
		60x30	35.78	38.28	40.96
		70x35	42.74	45.73	48.93
5	Double angle on same side without Lip	50x50	42.74	45.73	48.93
		60x60	45.47	48.65	52.06
		70x70	46.74	50.01	53.51
		50x25	30.01	32.11	34.36
		60x30	33.47	35.81	38.32
		70x35	39.78	42.56	45.54
6	Double angle on same side with Lip	50x50	43.41	46.45	49.70
		60x60	45.16	48.32	51.70
		70x70	49.41	52.87	56.57
				38.24	40.92
		50x25	35.74	46.77	50.04
		60x30	43.71	46.19	49.43
		70x35	43.17	37.94	40.60

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.

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.

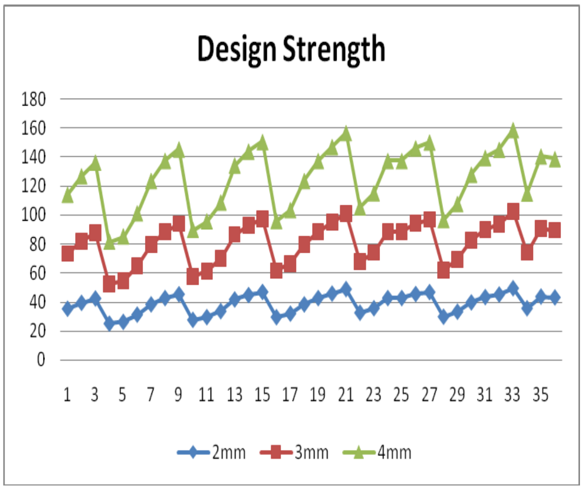

Fig 2Comparison of Design valves

COMPARISON OF EXPERIMENTAL AND NUMERICAL INVESTIGATION

The stress distributions obtained using ANSYS closely agrees with the experimental results within the elastic limit.

7.1 Comparison of Numerical valve in Ansys

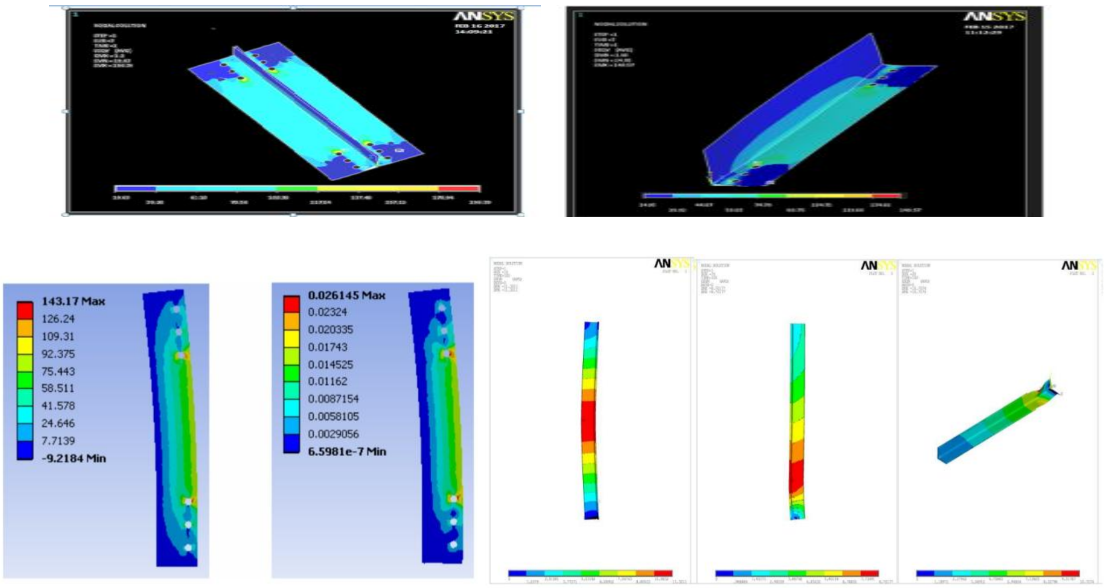

Fig 3Single angle without Lip , Displacement for 70mmx70mmx2mm

Fig 4 Double angle on same side without Lip Displacement for 60mmx60mmx2mm.

CONCLUSIONS

Based on the experimental, numerical and analytical results were concluded.

1. All angles section values predicted by the international codes BIS, AISI, AS/NZS and BS. Experimental Ultimate loads are nearly 10% to 12% less than the all codal provisions.
2. 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.

3. The proposed equation for net section tension capacity is applicable for all the section which gives more accruable value when compared to experimental values.

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, D.C.
5. AISI (2001). *Cold-Formed Steel Standard*, American Iron and Steel Institute, Washington, D.C.
6. ANSYS Release 8.1 (2004). ANSYS, Inc., USA
7. AS/NZS (1996). *Australian/New Zealand Standard*, Cold- Formed Steel Structures, Sydney, Australia.
8. Desmond, T. P., Pekoz, T., and Winter, G. (1981). "Edge stiffeners for thin-walled members." *Journal of Structural Engineering*, 107(2), pp. 329–353.

How to cite this article: Makesh AP, Arivalagan S. Structural performance of cold formed steel angle. International Journal of Research and Review. 2018; 5(12):203-206.

******