International Journal of Research in Engineering, Science and Management
Volume-1, Issue-8, August 2018
www.ijresm.com ISSN (Online): 2581-5782

Numerical Analysis of Cold Formed Steel Angle Section Members under Tension Load

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

[1]*Research Scholar, Dept. of Civil Engineering, Dr. M. G. R. Educational and Research Institute University, Chennai, India*

[2]*Professor and Dean, Dept. of Civil Engineering, Dr. M. G. R. Educational and Research Institute University, 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 1.5mm and 1.6mm and double angles sections of 1.5mm and 1.6mm under plain (without Lipped) and with Lipped conditions subjected to tension. Comparisons were made between the test results and the predictions based on both the Experimental investigation and the ANSYS analysis.**

Keywords: ANSYS, FEM, Work bench, shell.

I. Introduction

The finite element method is an extremely useful tool of analysis in many fields of Engineering. Finite element analysis is capable of giving a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material. Structural analysis can be carried out using linear and non-linear models. Linear models use simple parameters and assume that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities. The Finite Element Analysis was performed using the commercial finite element program ANSYS 16.2. The study compared the ultimate load carrying capacity of the single and double angle section from the FEM analysis with measured failure load from tension load. A non-linear analysis was performed and the materials are assumed to behave as an isotropic hardening material.

II. Finite Element Analysis

In this section, Finite element modelling of the experimental angle specimens is described. FEA as applied in engineering is a computational tool for performing engineering analysis. The FEA is performed using 3D structural solid elements that are capable of representing large deformation geometric and material non-linear. In the current study, each of the angle specimens is analysed. SOLID 185 is used for the 3D modelling of solid structures.

It is defined by eight nodes having three degrees of freedom at each node has translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials. The static material properties obtained from the tension tests and the measured cross-section dimensions were used to model the angle specimens.are unavoidable

A. About ANSYS Workbench

ANSYS workbench capabilities include a unique and extensive materials and sections for steel structures. In addition, the user could introduce the shapes or materials into the corresponding ANSYS workbench library. A user friendly beam and shell post processor includes listing the plotting section geometry and stresses and strain inside the cross section. The skilled combination module, select loads and coefficient for logic code combinations at element and global as well as worst load arrangement in shell and solid elements.

B. Modeling of Structures

Modeling is one of the most important aspects for the FEM analysis. Accuracy in the modeling of element type and size, geometry, material properties, boundary conditions and loads are absolutely necessary for close numerical idealization of the actual member. In the finite element model, the shear deformation of the bolts was ignored.

Finite element analysis consists of a computer model of a material or design that is stressed and analyzed for specific results. Finite element analysis uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes.

C. Element Types used for Modelling

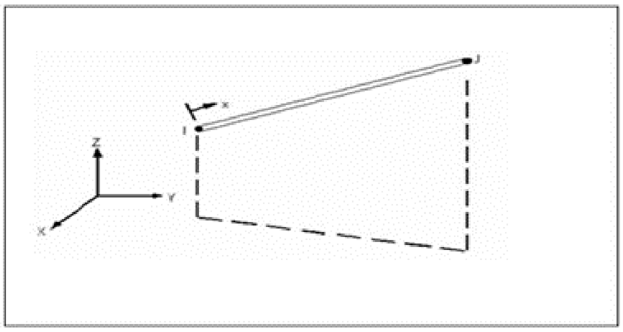

Fig. 1. Element characteristics of Shell 63

International Journal of Research in Engineering, Science and Management
Volume-1, Issue-8, August 2018
www.ijresm.com ISSN (Online): 2581-5782

SHELL 63 element types were used to model the single and double angle specimens. It is a 4 nodded 3-dimensional quadratic elastic shell element. It has both bending and membrane capabilities. This element has six degrees of freedom at each node: translations in the nodal x, y and z directions and rotations about the nodal x, y and z axes. Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection analysis. The Fig. 1 shows the element characteristics of SHELL 63.

D. Loading and Boundary Conditions

The end boundary conditions for all the finite element models were chosen to simulate the actual experimental set up. The bolted end conditions were considered as pinned end condition. For both single and double angle members the full length of the specimen was modeled.

In the finite element model, the shear deformation of the bolts was ignored. The load was assumed to transfer from the gusset plate to the angle fully by the bearing of the bolts. Therefore, one half of the circumference of each bolt hole in the model, which was supposed to bear against the bolt in the tests was fixed in the x and y translational degrees of freedom. Since the bolts have been tightened before loading and the bolts were still tight after tests, all nodes in the first two circumferences around the bolt holes of the specimen were fixed in the z translational degrees of freedom.

III. Literature Review

Kulak and Wu (1997) conducted a finite element analysis to evaluate the stress distribution of the critical cross section at ultimate load. A large strain four-node quadrilateral shell element with six degrees of freedom per node was used in the finite element modeling of the double angle members. The gusset plate was modeled using elastic four–node quadrilateral shell element as yielding of the gusset plate was not observed in the experimental tests. At failure, significant necking of the net area between the leg edge and lead bolt hole was observed.

Epstein and Chamarajanagar (1996) developed analytical model for a series of single angle tests with staggered bolted connections. A 20 node brick element was used in the finite element modeling of the angle sections to capture the stress concentration effect in the vicinity of bolt holes. The material nonlinear effects were modeled using the von Misses yield criterion and the material stress-strain curve was assumed to be elastic–perfectly plastic. In this study, a strain based failure criterion in which failure was assumed to have occurred once the maximum strain reached five times the initial yield strain was employed to capture the failure load.

Epstein McGinnis (2000) conducted a second study aimed at refining the tools developed in Epstein's 1996 work. The boundary conditions and the solution procedure were identical to the 1996 Epstein study. Although this finite element study included only the material nonlinearity as represented by a simple elastic-perfectly-plastic yield criterion, the finite element results indicated a reasonably good correlation with the experimental results.

Chung and Ip (2000) investigated the finite element modeling of bolted connections between cold-formed steel strips and hot-rolled steel plates under shear. The modeling was done with three-dimensional solid elements using the results of the coupon tests. Twelve lap shear tests with two steel grades, one bolt diameter and two washer sizes were carried out to caliber the finite element models. Typical strain levels in cold-formed steel strips in the vicinity of bolt holes were found to be 40%. Therefore it is important to incorporate reduced strength at larger strains for accurate prediction of the load-carrying capacities of bolted connections.

Gupta Mohan and Gupta (2004) conducted finite element analysis to evaluate the stress distribution in the angle at design loads predicted by equations developed earlier on the basis of experimental results. Detailed finite element analysis was conducted on three bolted angle specimens. These three angle specimens had two, three and four bolts at each end respectively. The resulting stress distribution justified the use of area along the gross shear plane in block shear strength prediction equation. The distribution and concentration of von Misses stresses indicated that block shear failure might occur in a two bolt connection, and net section failure might occur in three and four bolts connection.

IV. Numerical Investigation

ANSYS software was utilized to calculate the strength behavior of cold formed steel angle under tension load.

A. Non-Linear Analysis

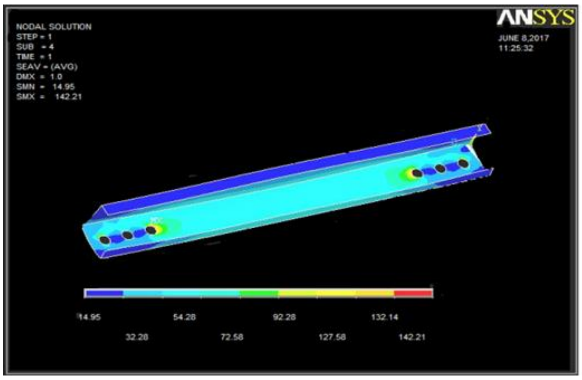

Fig. 2. Stress distribution for Single angle with Lip 50x50x2

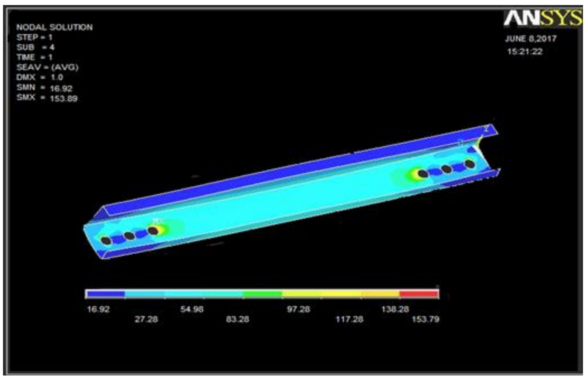

Fig. 3. Stress distribution for single with Lip 60 x60x2

To perform the non-linear analysis, the single and double

International Journal of Research in Engineering, Science and Management
Volume-1, Issue-8, August 2018
www.ijresm.com ISSN (Online): 2581-5782

angle specimens are modeled based on the experimental set up incorporating geometric imperfections. As the nonlinear problem is path dependent, the solution process requires a step by step load incremental analysis. In the analysis, the solution usually converged very slowly after yielding, and the increment for each load step had to be made very small.

B. Load vs. Deflection

The Fig. 4, shows the typical load versus deflection behavior for single angles with and without lips and double angles. From the graphs, it is observed that the ultimate load carrying capacity increases as the cross-sectional area and number of bolts in the connection increases.

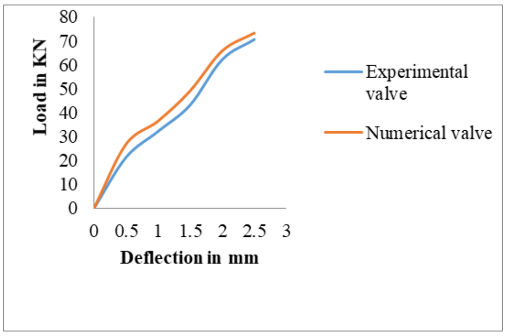

Fig. 4. Load vs. Deflection of single plain angle specimen

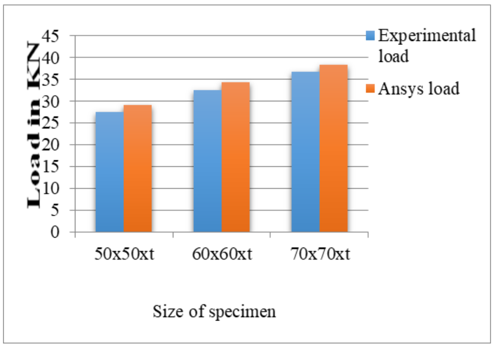

Fig. 5. Experimental load and ANSYS load for single plane angles (1.5mm)

C. Ultimate Load Carrying Capacity using ANSYS 16.2

TABLE I
COMPARISON OF EXPERIMENTAL LOAD AND NUMERICAL LOAD OF THICKNESS 1.5MM

S. No.	Description	Size of specimen (mm)	Exp. load (KN)	Ansys load (KN)	% increase in load
1	**Equal size**	50x50xt	29.45	31.12	5.67
2	Single angle without Lip	60x60xt	34.56	36.41	5.35
3		70x70xt	40.58	42.89	5.69
4	Single angle with Lip	50x50x10xt	39.15	41.28	5.44
5		60x60x10xt	46.78	48.78	4.28
6		70x70x10xt	52.58	54.89	4.39
7	Double angle opposite side without Lip	50x50xt	62.58	65.98	5.43
8		60x60xt	68.41	71.28	4.20
9		70x70xt	84.59	88.47	4.59
10	Double angle same side without Lip	50x50xt	62.58	65.21	4.20
11		60x60xt	76.24	79.34	4.07
12		70x70xt	84.25	87.87	4.30
13	Double angle opposite side with Lip	50x50x10xt	76.28	79.89	4.73
14		60x60x10xt	82.58	86.45	4.69
15		70x70x10xt	103.56	108.78	5.04
16	Double angle same side with Lip	50x50x10xt	76.42	79.89	4.54
17		60x60x10xt	88.48	92.89	4.98
18		70x70x10xt	106.58	111.28	4.41
19	**Unequal size**	50x25xt	21.58	22.75	5.42
20	Single angle without Lip	60x30xt	25.46	26.91	5.70
21		70x35xt	32.45	34.12	5.15
22	Single angle with Lip	50x25x10xt	28.11	29.71	5.69
23		60x30x10xt	31.44	32.76	4.20
24		70x35x10xt	39.58	41.75	5.48
25	Double angle opposite side without Lip	50x25xt	40.48	42.79	5.71
26		60x30xt	57.48	60.48	5.22
27		70x35xt	67.41	70.13	4.04
28	Double angle same side without Lip	50x25xt	41.33	43.51	5.27
29		60x30xt	52.58	55.18	4.94
30		70x35xt	68.47	72.26	5.54
31	Double angle opposite side with Lip	50x25x10xt	58.45	61.81	5.75
32		60x30x10xt	68.45	72.29	5.61
33		70x35x10xt	81.54	86.28	5.81
34	Double angle same side with Lip	50x25x10xt	56.18	59.13	5.25
35		60x30x10xt	67.28	71.28	5.95
36		70x35x10xt	81.59	85.89	5.27

TABLE II
COMPARISON OF EXPERIMENTAL LOAD AND NUMERICAL LOAD OF THICKNESS 1.5MM

S. No.	Description	Size of specimen (mm)	Exp. load (kN)	Ansys load (kN)	% increase in load
1	**Equal size**	50x50xt	27.54	29.12	5.74
2	Single angle without Lip	60x60xt	32.45	34.28	5.64
3		70x70xt	36.75	38.32	4.27
4	Single angle with Lip	50x50x10xt	36.28	37.98	4.69
5		60x60x10xt	42.58	44.32	4.09
6		70x70x10xt	48.56	51.28	5.60
7	Double angle opposite side without Lip	50x50xt	59.78	63.25	5.80
8		60x60xt	64.58	68.15	5.53
9		70x70xt	79.86	83.78	4.91
10	Double angle same side without Lip	50x50xt	56.78	59.42	4.65
11		60x60xt	64.58	68.18	5.57
12		70x70xt	78.54	82.37	4.88
13	Double angle opposite side with Lip	50x50x10xt	69.74	72.87	4.49
14		60x60x10xt	74.58	76.89	3.10
15		70x70x10xt	97.87	102.72	4.96

16	Double angle same side with Lip	50x50x10xt	68.74	71.82	4.48
17		60x60x10xt	80.47	83.51	3.78
18		70x70x10xt	96.47	99.72	3.37
19	**Unequal size** Single angle without Lip	50x25xt	18.27	19.28	5.53
20		60x30xt	22.47	23.81	5.96
21		70x35xt	28.47	30.12	5.80
22	Single angle with Lip	50x25x10xt	23.47	24.58	4.73
23		60x30x10xt	30.79	32.18	4.51
24		70x35x10xt	33.48	35.28	5.38
25	Double angle opposite side without Lip	50x25xt	38.78	40.91	5.49
26		60x30xt	49.78	51.89	4.24
27		70x35xt	58.47	61.29	4.82
28	Double angle same side without Lip	50x25xt	37.48	39.72	5.98
29		60x30xt	49.72	52.41	5.41
30		70x35xt	58.78	61.29	4.27
31	Double angle opposite side with Lip	50x25x10xt	50.43	52.38	3.87
32		60x30x10xt	54.58	56.27	3.10
33		70x35x10xt	70.59	73.45	4.05
34	Double angle same side with Lip	50x25x10xt	51.58	53.48	3.68
35		60x30x10xt	60.72	63.72	4.94
36		70x35x10xt	70.58	73.82	4.59

V. Conclusion

Numerical results shown that the ultimate strength of single equal angle lipped section under tension load is increase 1.24 times greater than single equal plain angle section. In the case of single unequal angle lipped section under tension load is increase 1.22 times greater than single unequal plain angle section. To examine that the ultimate strength of Double equal angle lipped section of opposite side under tension load is increase 1.26 times greater than of double equal angle plain section of opposite side. In the case of Double unequal angle lipped section of opposite side under tension load is increase 1.23 times greater than double unequal plain angle section of opposite side.

ANSYS software was utilized to calculate the strength behavior of cold formed steel angle under tension load. The numerical model developed using ANSYS to predict the behavior of single and double angles was found to simulate the experimental valves are closely.

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.Google Scholar

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

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

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

[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.Google Scholar

[9] Lam, S. S. E., Chung, K. F., and Wang, X. P. (2006). “Load carrying capacities of cold-formed steel cut stub columns with lipped C-section.” Thin-Walled Structures, 44, pp. 1077–1083.Google Scholar

[10] NAS (2001). Specification for the Design of Cold-formed Steel Structural Members, North American Specification, Washington, D.C. Google Scholar.