Chief Editor
Dr. A. Singaraj, M.A., M.Phil., Ph.D.
Editor
Mrs.M.Josephin Immaculate Ruba

EDITORIAL ADVISORS

1. Prof. Dr.Said I.Shalaby, MD,Ph.D.
   Professor & Vice President
   Tropical Medicine,
   Hepatology & Gastroenterology, NRC,
   Academy of Scientific Research and Technology,
   Cairo, Egypt.
2. Dr. Mussie T. Tessema,
   Associate Professor,
   Department of Business Administration,
   Winona State University, MN,
   United States of America,
3. Dr. Mengsteab Tesfayohannes,
   Associate Professor,
   Department of Management,
   Sigmund Weis School of Business,
   Susquehanna University,
   Selinsgrove, PENN,
   United States of America,
4. Dr. Ahmed Sebihi
   Associate Professor
   Islamic Culture and Social Sciences (ICSS),
   Department of General Education (DGE),
   Gulf Medical University (GMU),
   UAE.
5. Dr. Anne Maduka,
   Assistant Professor,
   Department of Economics,
   Anambra State University,
   Igbariam Campus,
   Nigeria.
6. Dr. D.K. Awasthi, M.SC., Ph.D.
   Associate Professor
   Department of Chemistry,
   Sri J.N.P.G. College,
   Charbagh, Lucknow,
   Uttar Pradesh. India
7. Dr. Tirtharaj Bhoi, M.A, Ph.D,
   Assistant Professor,
   School of Social Science,
   University of Jammu,
   Jammu, Jammu & Kashmir, India.
8. Dr. Pradeep Kumar Choudhury,
   Assistant Professor,
   Institute for Studies in Industrial Development,
   An ICSSR Research Institute,
   New Delhi- 110070, India.
9. Dr. Gyanendra Awasthi, M.Sc., Ph.D., NET
   Associate Professor & HOD
   Department of Biochemistry,
   Dolphin (PG) Institute of Biomedical & Natural Sciences,
   Dehradun, Uttarakhand, India.
10. Dr. C. Satapathy,
    Director,
    Amity Humanity Foundation,
    Amity Business School, Bhubaneswar,
    Orissa, India.

ISSN (Online): 2455-7838
SJIF Impact Factor (2017): 5.705

EPRA International Journal of Research & Development (IJRD)

Monthly Peer Reviewed & Indexed
International Online Journal

Volume: 3, Issue:10, October 2018

Published By :
EPRA Journals

CC License
BY NC ND

SJIF Impact Factor: 5.705 Volume: 3 | Issue: 10 |October| 2018 ISSN: 2455-7838(Online)
EPRA International Journal of Research and Development (IJRD)

NUMERICAL INVESTIGATION OF COLD FORMED STEEL ANGLE SECTIONS USING ANSYS WORKBENCH UNDER TENSILE LOADING CONDITIONS

Paul Makesh A[1]

[1]PhD Research Scholar, Department of Civil Engineering, Dr.M.G.R Educational and Research Institute University, Maduravoyal, Chennai -600095, Tamilnadu, India.

Arivalagan S[2]

[2]Professor and Dean, Department of Civil Engineering, Dr.M.G.R Educational and Research Institute University, Maduravoyal, Chennai-600095, India.

ABSTRACT

The finite element method is an extremely useful tool of analysis in many fields of Engineering. Finite Element Analysis is an accurate and flexible technique to forecast the performance of a structure, mechanism or process under different loading conditions. This work presents 72 specimens carried out on tension members fastened with bolts, to calculate the ultimate strength on cold-formed steel using ANSYS workbench. This analysis carries single angle sections and double angles sections under tensile condition of thickness 1.5mm and 1.6mm Models were been developed to calculate the ultimate strength of single and double sections from linear through nonlinear response and up to failure. Modeling simplifications and assumptions developed during this research are presented. The study compared the ultimate load carrying capacity of the single and double angle section from the finite element analysis with measured failure load from tension loads. The Static Structural component and all of its modules is created. The modules are similar to those in ANSYS MAPDL (Mechanical Ansys Parametric Design Language).The static analysis of FEA are Build geometry, define materials, Generate mesh, apply load, solving the model, and reviewing the results.

KEYWORDS: *ANSYS workbench, MAPDL,Meshing, Loading , Element, FEM*

INTRODUCTION

Finite element analysis of cold-formed structures plays an increasingly important role in engineering problems, as it is relatively inexpensive and time efficient compared to physical experiments, especially when a parametric study of cross section geometries is involved. FEA is an efficient analytical tool, which can predict the behaviour accurately, provided the model developed resembles the experimental prototype. Hence, it is necessary to verify the finite element results with the experimental results. In general, finite element analysis is a powerful tool in predicting the ultimate loads and complex failure modes of cold-formed structural members

Finite Element Analysis

CivilFEM is a set of preprocessing, solution and post processing tools that is integrated within ANSYS and makes it easier for the user to deal with civil engineering problems. The goal of the finite element analysis is to develop a model that could study the behavior of bolted cold-formed steel single and double angle tension members. The behavior observed during the tests was used for preparing a finite element model, particularly during the non-linear analysis. In angles under tension, the behavior is highly nonlinear as the failure approaches.

ANSYS Workbench

ANSYS Mechanical (Workbench) 16.0 makes it easy to review element quality during the development of meshing controls, as well as in post processing results. Well-shaped elements yield superior results, and help reduce element shape errors during large displacement analysis, such as when using hyperelastic materials with substantial strain. This article briefly reviews features in the Workbench Mechanical Outline for displaying element quality (1) in color in the Mesh branch, (2) using bar charts of element quality, and (3) plotting element quality in post processing after element shapes have been distorted by strain in a model.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.

Static Structural Analysis

A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. A static structural load was performed using the ANSYS 15.The static material properties obtained from the tension tests and the measured cross-section dimensions were used to model the angle specimens. 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. A non-linear analysis was performed and the materials are assumed to behave as an isotropic hardening material. In this chapter, performing linear static structural analyses in Simulation are follows.

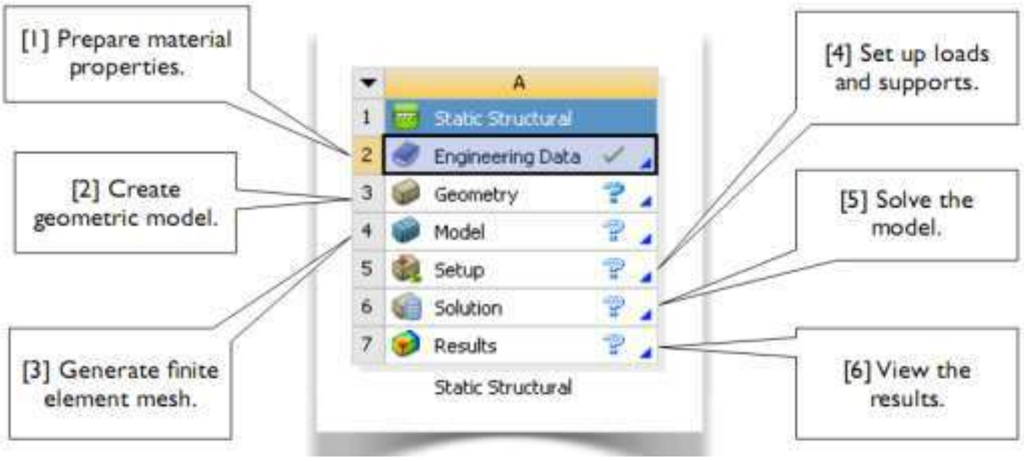

Fig 1 Static Structural Simulations

Modeling of Structures

To study the numerical investigations of cold formed steel single and double angle members under tension loading using the finite element analysis software ANSYS 15.0 was used to obtain a prediction of the ultimate load and evaluate the shear deformation at the cross section. 72 specimens were investigated using ANSYS of the thickness 1.5mm, 1.6mm, was used. All Angle specimens were connected with their larger leg to end gusset plates of mild steel of 8mm thickness using 10mm bolts. All the specimens were fabricated for a length of 500mm. The size of gusset plates are 70mm x 280mm for single angles specimen and double angles specimen on opposite side, similarly the size of gusset plates are 150mm x 280mm for double angle specimen on same side. Fig 1 2 shows the modeling of single angle sections and double angle sections.

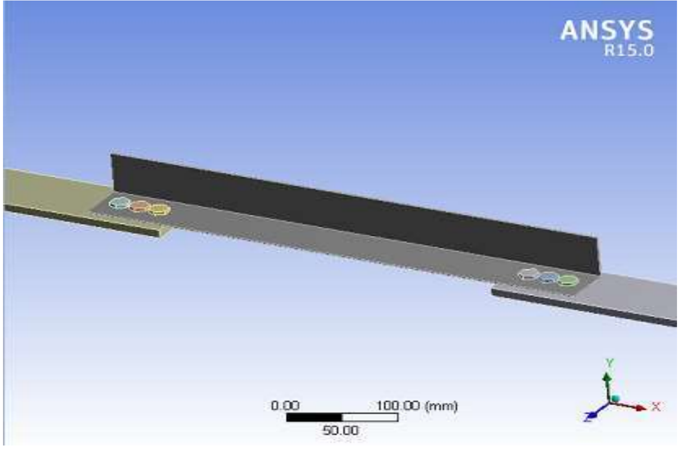

Fig 2 Single angle without lip 50x50x1.5

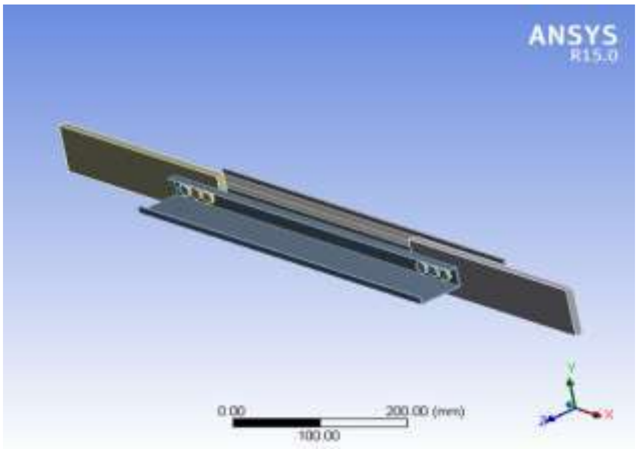

Fig 3 Double angle opposide side with lip 60x30x1.6

Element co ordinates system

The ANSYS program has several types of coordinate systems, each used for a different reason. Global and local coordinate systems are used to locate geometry in space. The display coordinate system determines the system in which geometry items are listed. The nodal coordinate system defines the degree of freedom directions at each node and the orientation of nodal results data. The element coordinate system determines the orientation of material properties and element results data.

Element Type

Geometry, loading and required results all need to be evaluated as part of the element selection process. The ANSYS program has a large library of element types. Some of the characteristics of the element types, and their groupings, are described in this chapter to make element type selection easier. CONTRA 174, SOLID 186, SOILD 187, SURF 175 and TARGE 170 are element types was used to model the single angle sections and double angle sections

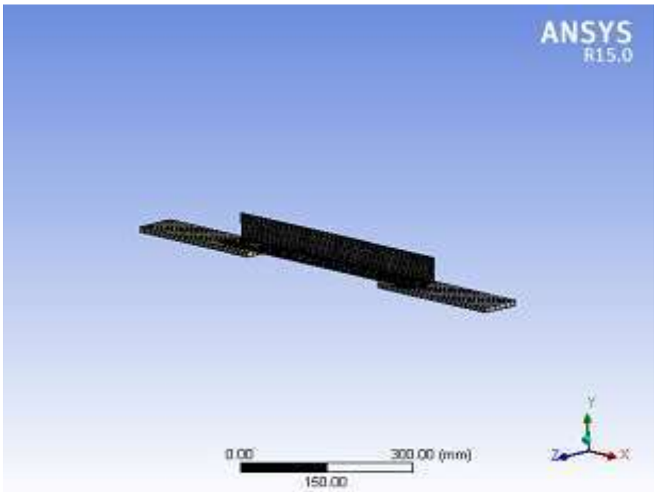

Fig 4 Single angle without lip 50x50x1.5

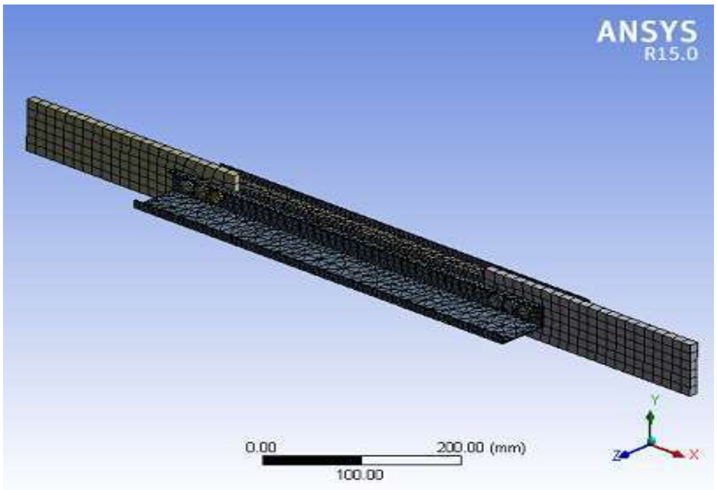

Fig 5 Double angle opposide side with lip 60x30x1.6

Loading and boundary conditions

The finite element model is created by using ANSYS workbench 15 software. Element types are used to mesh the geometry of the specimens. The boundary conditions for all the specimens were chosen to simulate the actual experimental set up. The lengths of specimens are 500mm.

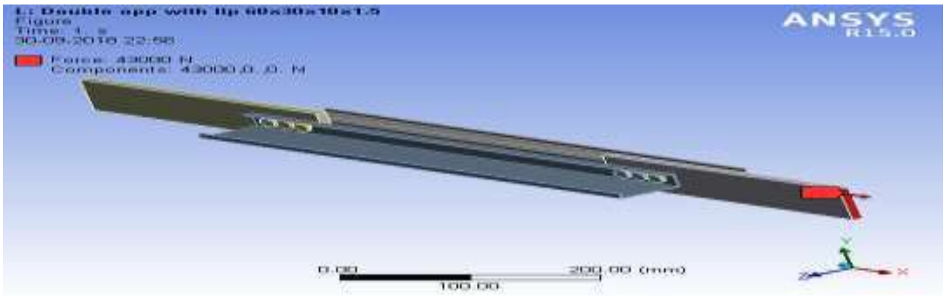

Fig 6 Loading conditions of double angle opposide side with lip 60x30x10x1.6

RESULT AND DISCUSSION

To perform the non-linear analysis, the single and double angle specimens are modeled based on the experimental set up incorporating geometric imperfections. As the nonlinear problem is path dependant, 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. The geometric imperfections included the thickness of the section, width of the connected leg, width of unconnected leg in case of single plain angles and it includes width of lip in case of lipped angles. Yielding is determined using von-Mises yield criteria.

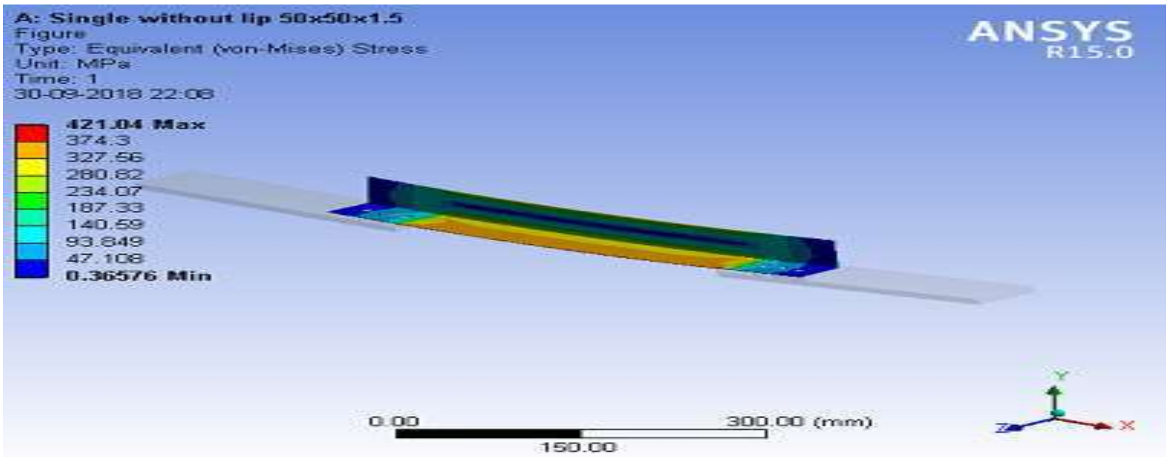

Fig 7 Stress distribution for single plane angle without lip 50x50x1.5

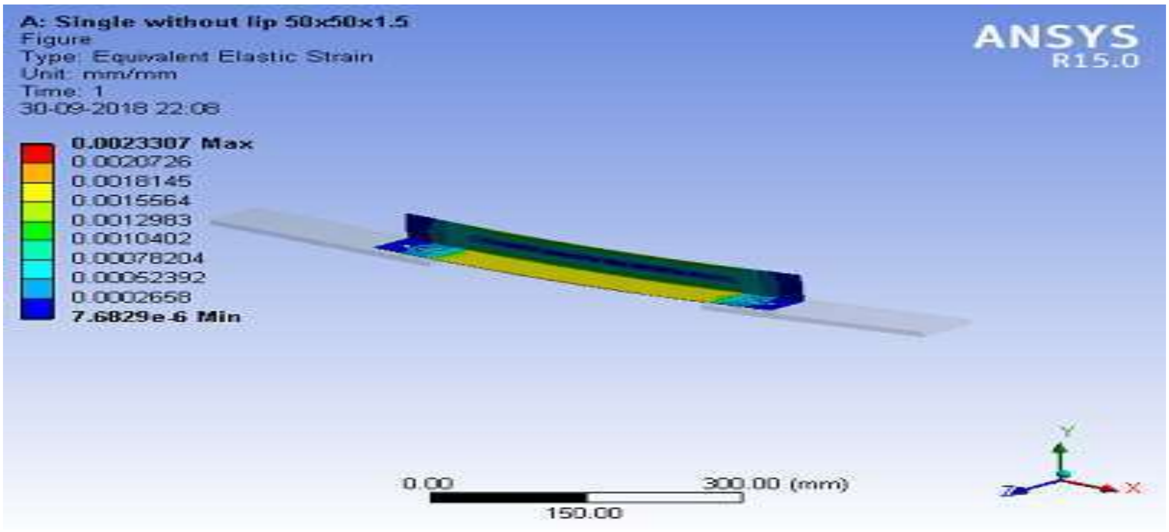

Fig 8 Strain distribution for single plane angle without lip 50x50x1.5

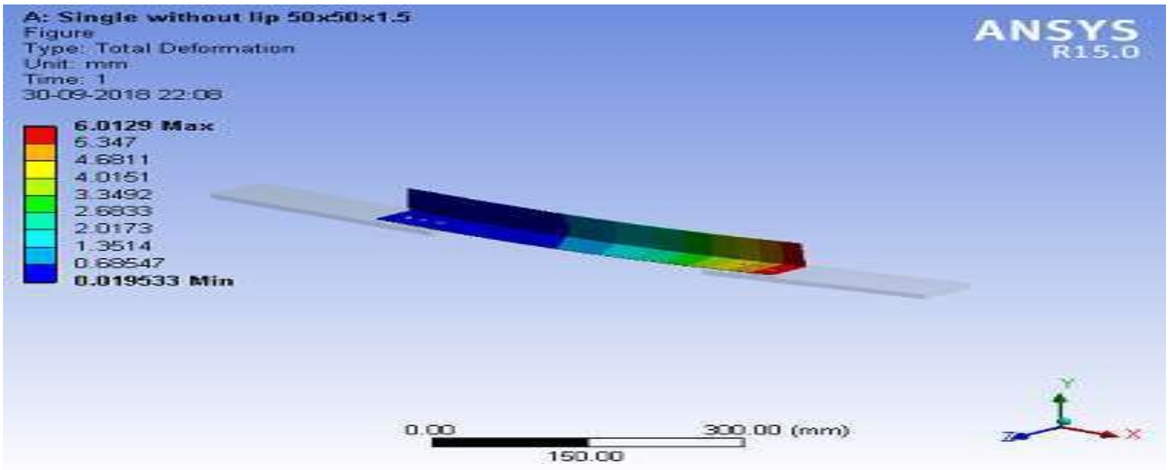

Fig 9 Total deformation for single plane angle without lip 50x50x1.5

.

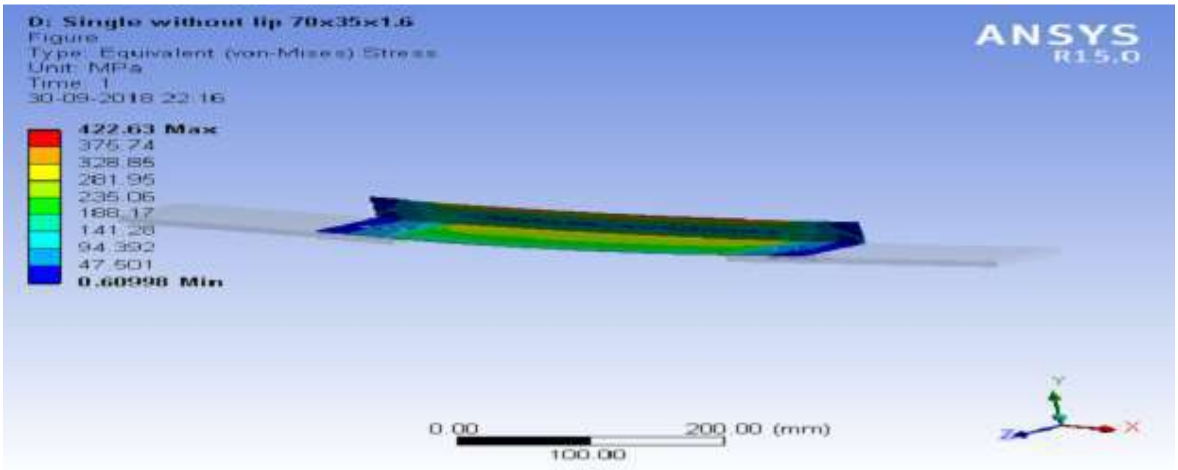

Fig 10 Stress distribution for single angle without lip 70x35x1.6

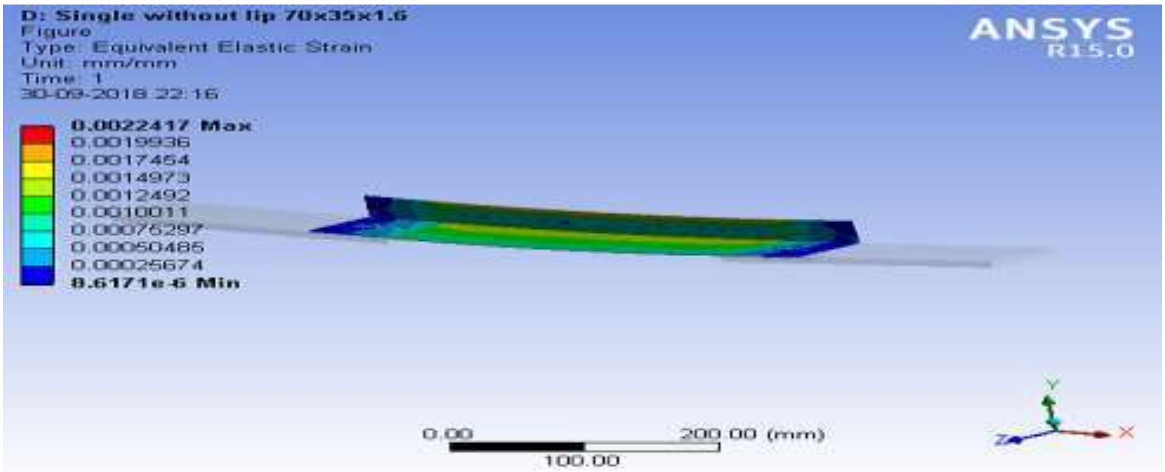

Fig 11 Strain distribution for single angle without lip 70x35x1.6

Table 1 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** Single angle without Lip	50x50xt	27.54	29.12	5.74
2		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 sidewithout 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

Table 2Comparison of Experimental load and Numerical load of thickness 1.6mm

S.No	Description	Size of specimen (mm)	Exp load (KN)	Ansys load (KN)	% increase in load
1	**Equal size** Single angle without Lip	50x50xt	29.45	31.12	5.67
2		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** Single angle without Lip	50x25xt	21.58	22.75	5.42
20		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

CONCLUSION

Numerical results shown that the ultimate strength of single equal plain angle sections without lip 5% higher than the experimental loads under tension. 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. 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.

REFERENCES

1. *Chung, KR & Ip, KH 2000, 'Finite Element Modeling of Bolted Connections between cold formed steel strips and hot rolled steel plates under static shear loading', Engineering Structures, vol. 4, no. 22, pp. 1271-1284*
2. *Gotluru 2000, 'Behavior of cold formed steel beams having open sections which were subjected to torsion', Journal of Engineering structures, vol. 2, no. 3, pp. 1374-1975*

3. *Hancock, GJ 2003, 'Cold-formed Steel Structures', Journal of Constructional Steel Research, vol. 59, no. 4, pp. 473-487*
4. *Kulak, GL & Grondin, GY 2001, 'AISC Rules for Block Shear in Bolted Connections', Engineering Journal, vol. 5, no. 4, pp. 199-203*
5. *Young & Hancock 2012, 'Experimentally investigated the cold formed steel channel section to combined bending and web crippling', Journal of cold formed steel structures, vol. 12, no. 11, pp. 105-112*
6. *Schifferaw & Schafer 2007, 'Inelastic bending capacity in cold formed steel members', Construction research of steel structures, vol. 6, no. 5, pp. 571-553*
7. *Sonal Banchhor 2016, 'Behavior of cold formed steel bolted angle tension members', International journal of constructional steel Research, vol. 15, no. 6, pp. 751-764*