International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 5.22 (SJIF-2017), e-ISSN: 2455-2585
Volume 4, Issue 09, September-2018

FEM ANALYSIS OF PERFORATED STEEL EQUAL ANGLE MEMBERS

M.Jayakumar Bhaskar[1] ,SK.Rahaman[2]

[1]*Assistant Professor,Civil engineering department,Visvodaya engineering college,kavali,india.*
[2]*M.Tech scholor,Civil engineering department,Visvodaya engineering college,kavali,india.*

Abstract—**Hot-rolled steel angle sections are usually used in many of the structural applications, for example industrial plants, lattice power transmission towers, where they are used as supporting structures to pipes and elevators, roof trusses, shelves, overhead tanks, chimneys, bunkers, silos and storage pallet racks etc. They are also frequently used in both roof braces and lateral braces of industrial structures.**

"This paper is concerned with the ultimate load capacity of non-perforated and perforated channel hot rolled steel column. An analytical study has been undertaken to investigate the behaviour of such members under compression. FEM models are used in order to carry out buckling and non linear analyses with the aim of detecting the capacity of the channel members in case of one or more perforations.The software used for finite element analysis in this project is ABAQUS."

Keywords— imperfections,Angle member with perforations,one,two,three perforations.

I. INTRODUCTION

Hot-rolled steel angle sections are usually used in many of the structural applications, for example industrial plants, lattice power transmission towers, where they are used as supporting structures to pipes and elevators, roof trusses, shelves, overhead tanks, chimneys, bunkers, silos and storage pallet racks etc. They are also frequently used in both roof braces and lateral braces of industrial structures. In the previous decades many analytical and experimental research has been developed to the performance assessment of angle members subjected to compression. Many of the structural members are in compression. Compression members in buildings are called columns, stanchions or posts. Compression member in truss is called as strut. The jib of crane is known as boom which takes compression.

A. OBJECTIVE OF THE PROJECT

The main objective of this project is

- To develop a finite element (FE) model for assessing the axial behaviour of steel angle members with perforations in varying numbers such as single, double, triple perforations in both the legs of steel members.
- The software used for analyzing is 'ABAQUS'
-

B. SCOPE OF THE PROJECT

The scope of this project lies in understanding the behavior of the perforated steel angle members in consideration by performing analysis on the members. They are mentioned below

- Study is limited to single, double, triple perforations in both the legs of steel members.
- Only axial behaviour will be studied under static loading
- Study is limited to angle sections.

Design of the compression members is carried according to IS 800: 2007.

II.LITERATURE REVIEW

Wei-bin Yuan et.al (2017) determines the numerical and analytical investigations on the distortional buckling of perforated cold-formed steel channel-section beams with circular holes in internet. The numerical investigation involves the utilization of finite component ways. The influence of the net holes on the distortional buckling behaviour and corresponding important stress and moment of perforated cold-formed steel channel-section beams ar mentioned. Finally, an easy analytical formulation is projected for evaluating the impact of hole size on the reduction of important stress and important moment of the channel-section beams with circular holes in web.

Xingyou Yao et.al. (2013) Cold-formed steel structural sections used in the walls of residential buildings and agricultural facilities are commonly C-shaped sections with web holes. These holes located in the web of sections can alter the elastic stiffness and the ultimate strength of a structural member. The objective of this paper is to study the buckling mode and load-carrying capacity of cold-formed thin-walled steel column with slotted web holes.

Nabil Abdel-Rahman et.al. (2011) deals with experimental Perforations ar usually provided within the net and (or) rim plates of beams and columns of cold-formed steel (CFS) structural members so as to facilitate duct work, piping, and bridging. This paper thinks about with the institution of effective style breadth equations for the determination of the final word strength of such perforated members in compression. A established finite component model has been wont to study the consequences of perforation parameters on the final word strength of perforated members. The finite component model consists of short columns of liplike channel CFS sections, discretized exploitation nonlinear "assumed strain" shell finite components, and utilising experimental-based material properties models. The constant study covers net slenderness values between thirty one and 194, perforation breadth to net breadth ratios up to zero.6, and perforation height to perforation breadth ratios up to three.0. Effective style breadth equations for plates having sq. perforations and elongated perforations were developed. The potency and accuracy of those 2 equations in predicting the final word strength of perforated CFS compression members are verified through a comparison with the final word load results of many experimental studies from the literature.

Y.C Wanget.al. (2009)proposes cold-formed thin-walled steel sections ar wide used as primary load bearing members in light-weight panels that kind walls in residential and different low rise structures. In cold regions, the webs of the steel sections ar usually perforated cut back|to scale back|to cut back} the cold bridging result so as to extend thermal comfort and reduce energy waste. Perforating the online of a steel section can cut back its load bearing capability. This paper presents the results of Associate in Nursing experimental and numerical study to analyze the compression behavior of light-weight structural panels victimization perforated sections. the first objective of the tests is to supply experimental information to validate the numerical simulations, that were applied victimization the business finite component analysis computer code ABAQUS. The valid atomic number 26 analysis was wont to develop a straightforward style calculation methodology to convert a neighborhood with perforated internet to a neighborhood with solid internet. within the equivalent solid internet, the thickness of the solid internet would have constant elastic native buckling strength because the original perforated internet with the gross thickness whereas the thickness of the imperforated flanges remains unchanged. By changing a thin-walled section with perforated internet to a solid section with an efficient internet thickness, the standard style ways for thin-walled structures may be applied.

III METHODOLOGY

The various steps followed in this project is shown below

1. Literature survey
2. Modeling in ABAQUS
3. Parametric studies
4. Conclusion

As seen from the above methodology the first step involves collection of various literature's related to the current study. The next step is to collect experimental results from the collected literature's and studying their results. This can be achieved by using Limit State Design in accordance with IS: 800-2007. ISA 100 x 100 x 6 sections shall be considered for the analysis.

The main aim of this project is to calculate and compare the capacity and deflection values for analytically. A steel framed structural element is modeled using a Finite Element Analysis (FEA) Software i.e. ABAQUS software. The compressive load is considered in this project by calculating as per IS: 800-2007. The loads calculated are applied to the angle members using ABAQUS and the deflection values obtained are noted and tabulated.

IV FINITE ELEMENT MODELING AND ANALYSIS

A.General

Analytical and finite element analysis is always carried out to predict its behavior and capacity before carrying out the experiments to scrutinize number of tests so that large number of expensive experiments can be avoided. In the present investigation, a parametric study has been carried out using ABAQUS. Two types of analysis is performed, namely

1) Buckling analysis to incorporate the initial imperfections 2) Nonlinear post buckling analysis to predict the capacity, load deformation behavior and also for finding out the different buckling modes. The details of finite element modeling and analysis (FEA) are discussed in the following sections.

B. Modeling Description

Material non-linearity in the hot rolled steel angle can be modeled with Von Mises yield criteria. The steel angle members are modeled by bilinear material model, as linear elastic - perfectly plastic material; f_y= 250 N/mm^2; the Young's modulus E = 200000 N/mm^{2}and the Poisson's ratio μ = 0.3. At the ends of the element is defined as gusseted region which is arrested in vertical direction and in the bolted region on both ends of the member, fixed connection is applied as support. The plastic properties are given in the following table.

Yield Stress (N/mm^2)	Plastic Strain
200	0
246	0.0235
294	0.0474
374	0.0935
437	0.1377
480	0.18

Table 1 Plastic Properties

Finite element model for axial loading compression members with axial direction released is developed using ABAQUS. The member is loaded through one leg. The loading compression member has been modeled as shell element.

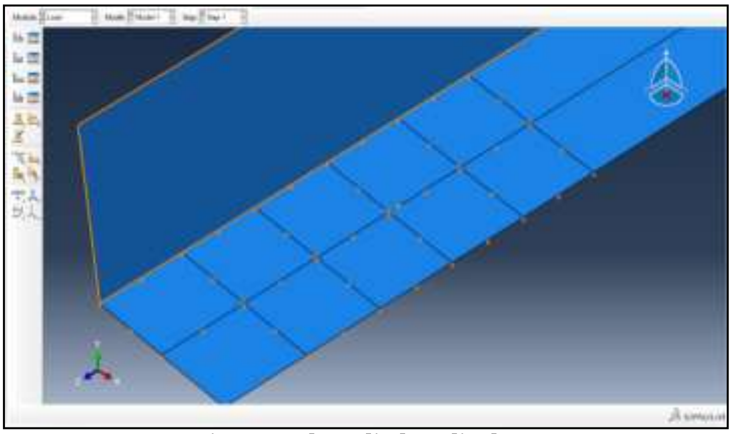

Fig 1 Load applied as displacement

C. Bundary Conditions

The boundary conditions are applied in such a way to simulate the real test setup. The experiments were conducted in loading frame, in which the both ends of the specimens is connected to gusset plate with bolts and is assigned with fixed boundary condition at the bolted region at one end and free at the other end as shown in figure 4.2. The top end of the specimen is assigned with fixed boundary condition, but released to move axially. The “load” in the non-linear finite element analysis was applied as the displacement at the third bolt at one end, negative 40mm displacement mimicking the loading method of the testing. Automatic load increment method is preferred because ABAQUS tool selects increment size based on computational efficiency.

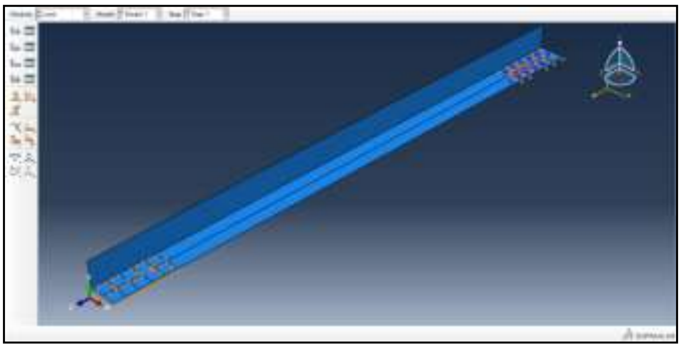

Fig 2 Boundary Condition

D. Analysis of buckling in Abaqus

The buckling analysis is carried to predict the buckling loads and the corresponding buckling shapes. These are used as a parameter in determining the post buckling strength and have additional application for incorporating the input values of the geometric imperfection using first buckling mode shape values.

- Initially angles are assigned material and cross sectional member properties and meshed.
- An additional step is created in ABAQUS. For buckling analysis; all boundary condition from initial step is propagated to this newly created step.

- For buckling analysis to take place an initial displacement of 1mm is applied at the required location on the section.
- The above created model is analyzed using Lanczos Eigen Solver requesting an Eigen value of fifty (50) to obtain overall mode of buckling.
- Form the obtained results of buckling analysis; over-all buckling is selected for incorporating the imperfection modeling.

E. Analysis and Results

On performing the analysis on the steel angle sections, deflection values are obtained which has been plotted and compiled in chapter **V.**

V RESULTS AND DISCUSSIONS

A.General

The results obtained after conducting experiment and performing FEM analysis for varying number of perforations and non-perforated angle members have been listed in detail.

B.Analytical Results

The angle members with varying in number of perforations and without perforations are modeled for length of 1500 mm, and 6mm thickness.

B.1 ISA 100 x 100 x 6 mm of Length 1500 mm

B.1.1 ***Deflection of the member without perforations before implementing imperfections***

Buckling analysis is carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 3 shows the buckling mode of the member before implementing the imperfection in ABAQUS.

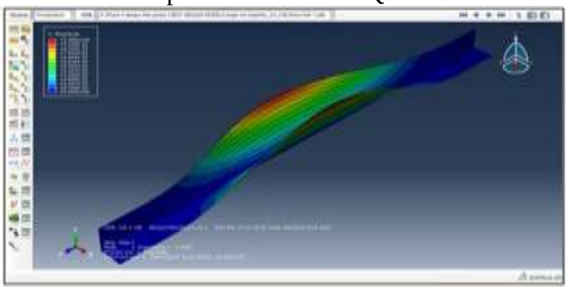

Fig 3 Buckling of the Angle Member without Perforations before implementing imperfections

B.1.2 ***Deflection of the member without perforations after implementing imperfections***

Buckling analysis of the member after imperfection is implemented into the Abaqus file. Figure.4 shows the buckling mode of the member after implementing the imperfection in ABAQUS.

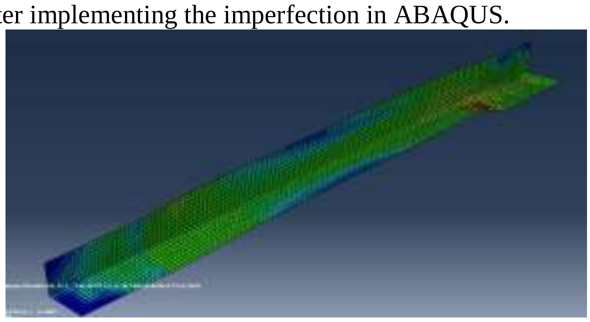

Fig 4 Buckling of the Angle Member without Perforations after implementing the imperfections

B.1.3 ***Deflection of the member with single perforation before implementing imperfections***

Buckling analysis is carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 5 shows the buckling mode of the perforated member before implementing the imperfection in ABAQUS.

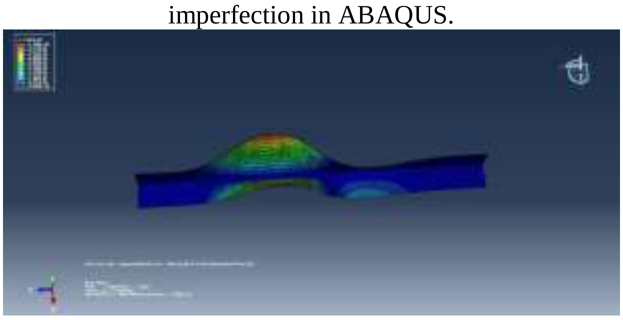

Fig 5 Buckling of the single perforation angle member before implementing imperfections

B.1.4 ***Deflection of the member with single perforations after implementing imperfections***

Buckling analysis of the member after imperfection is implemented into the Abaqus file. Figure.6 shows the buckling mode of the perforated member after implementing the imperfection in ABAQUS.

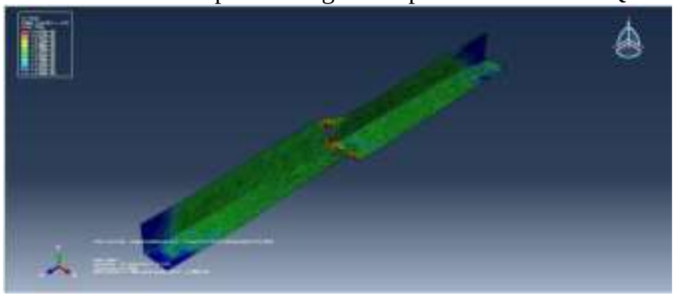

Fig 6 Buckling of the Angle Member with single perforations after implementing the imperfections

B.1.5 ***Deflection of the member with two perforations before implementing imperfections***

Buckling analysis is carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 7 shows the buckling mode of the perforated member before implementing the imperfection in ABAQUS.

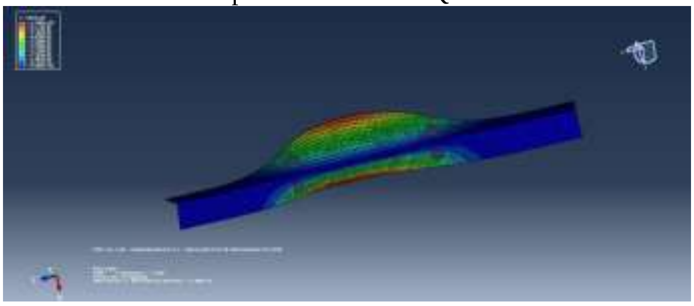

Fig 7 Buckling of the Angle Member with two Perforations before implementing the imperfections

B.1.6 ***Deflection of the member with two perforations after implementing imperfections***

Buckling analysis of the member after imperfection is implemented into the Abaqus file. Figure.8 shows the buckling mode of the perforated member after implementing the imperfection in ABAQUS.

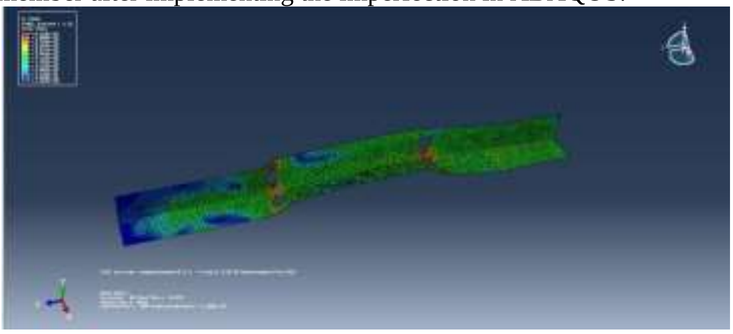

Fig 8 Buckling of the Angle Member with two Perforations after implementing the imperfections

B.1.7 ***Deflection of the member with three perforations before implementing imperfection***

Buckling analysis is carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 9 shows the buckling mode of the perforated member before implementing the imperfection in ABAQUS.

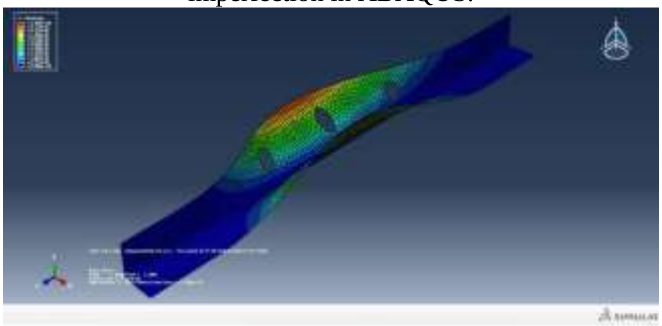

Fig 9 Buckling of the Angle Member with three Perforations before implementing the imperfections

B.1.8 ***Deflection of the member with three perforations after implementing material imperfection***

Buckling analysis of the member after imperfection is implemented into the Abaqus file. Figure 10 shows the buckling mode of the perforated member after implementing the imperfection in ABAQUS.

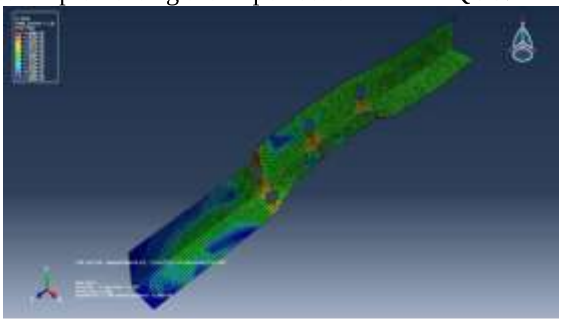

Fig 10 Buckling of the Angle Member with three Perforations after implementing the imperfections

B.1.9 Load vs. deflection graph for Angle Member without Perforations

The following figure 11 shows the capacity for angle member of 1.5m length.

The maximum load carrying capacity of the angle member ISA 100 x 100 x 6 mm is approximately 175 KN after that the load carrying capacity is gradually decreasing as shown in fig 11

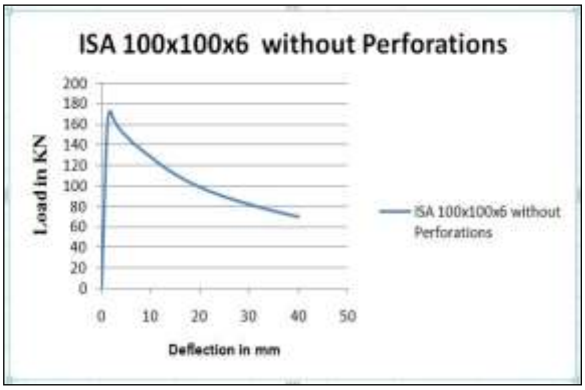

Fig 11 Load vs. Deflection graph for angle member without perforations

B.1.10 Load vs. deflection graph for Angle Member with Single Perforations

The following figure 12 shows the capacity reduction of the perforated angle member. The maximum load carrying capacity of the perforated angle member ISA 100 x 100 x 6 mm is approximately 145 KN after that the load carrying capacity is gradually decreasing as shown in fig 12

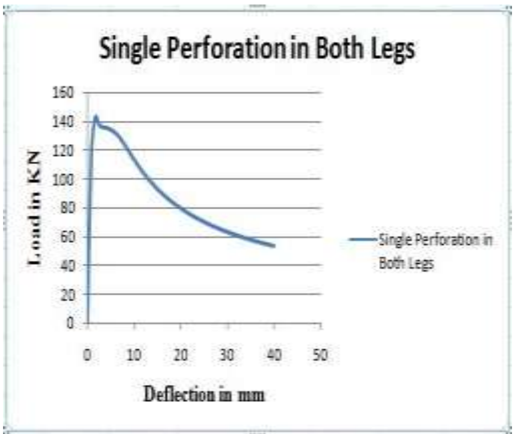

Fig 12 Load vs. Deflection graph for single perforated angle member

B.1.11 Load vs. deflection graph for Angle Member with Two Perforations

The following figure 13 shows the capacity reduction of the two perforations in angle member. The maximum load carrying capacity of the perforated angle member ISA 100 x 100 x 6 mm is approximately 130 KN after that the load carrying capacity is gradually decreasing as shown in fig 13.

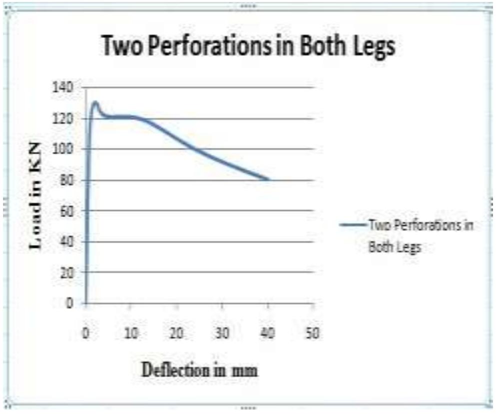

Fig 13 Load vs. Deflection graph for two perforated angle member

B.1.12 Load vs. deflection graph for Angle Member with Three Perforations

The following figure 14 shows the capacity reduction of the three perforations in angle member. The maximum load carrying capacity of the perforated angle member ISA 100 x 100 x 6 mm is approximately 130 KN after that the load carrying capacity is gradually decreasing as shown in fig 14.

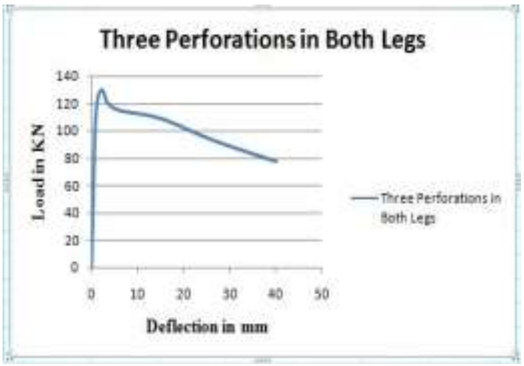

Fig 14 Load vs. Deflection graph for three perforations in angle member

B.1.13 Load vs. deflection graph Comparison of both Angle Members with single and non-perforated member.

The fig 15 represents a comparison of angle members with and non-perforated member.

1. From the following figure we can see that load carrying capacity of single perforated angle member is less than that of angel member Non-perforated member.
2. After the peak load we can see that for angle member Non-perforated, even for small increase in load there is more deformation.
3. For single perforated angle member after the peak load, further increase in load there is less deformation when compared to the angle member without perforations.

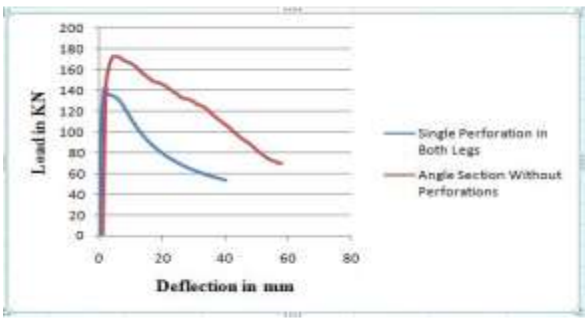

Fig 15 Load vs. Deflection graph for single perforations in angle member and non- perforated angle member

B.1.14 Load vs. deflection graph Comparison of both Angle Members with two perforations and non-perforated member.

The fig 16 represents a comparison of angle members with two perforations and non-perforated member.

From the following figure we can see that load carrying capacity of two perforated angle member is **less** than that of angel member with non-perforated member.

1. After the peak load we can see that for angle member with non-perforated member, even for small increase in load there is **more** deformation.
2. But for two perforated angle member after the peak load, further increase in load there is **less** deformation when compared to the angle member with non-perforated member.

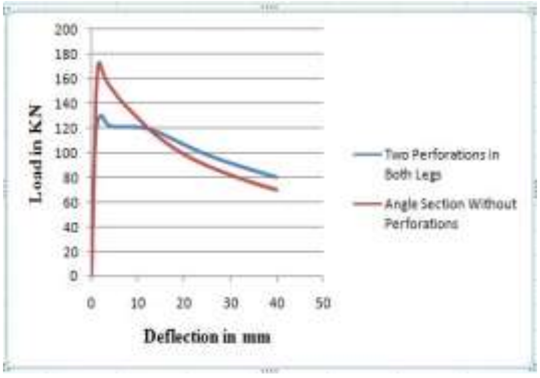

Fig 16 Load vs. Deflection graph for two perforations and non- perforated angle member

B.1.15 Load vs. deflection graph Comparison of both Angle Members with three perforations and non-perforated member.

The fig 17 represents a comparison of angle members with three perforations and non-perforated Member.

From the following figure we can see that load carrying capacity of three perforated angle member is **less** than that of angel member with non-perforated member.

1. After the peak load we can see that for angle member with non-perforated member, even for small increase in load there is **more** deformation.
2. But for three perforated angle member after the peak load, further increase in load there is **less** deformation when compared to the angle member with non-perforated member.

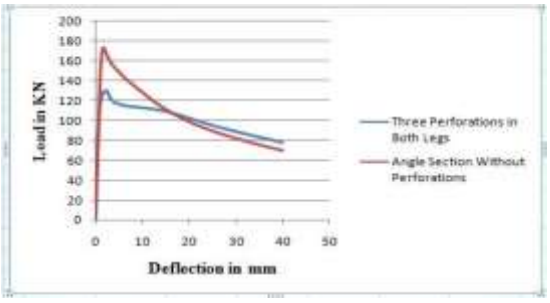

Fig 17 Load vs. Deflection graph for Three Perforated angle member and non- perforated angle member

C. Behavior Of Perforated And Non-Perforated Angle Members ISA 100x100x6 mm in ABAQUS

Sl.No	Pattern of Perforations	Abaqus	
		Capacity (KN)	Deflection (mm)
1.	Single Perforation in Both the Legs	145	2
2.	Two Perforations in Both the Legs	130	3
3.	Three Perforations in Both the Legs	120	2
4.	Without Perforations	175	3

VI CONCLUSIONS

"In the current study comprehensive analysis on Angle elements with and without perforations are carried out. The influence of the perforations on the capacity of the member and effect on behaviour of the buckling and resistance are investigated. In the study perforations is applied in increase in no. of perforations up to three perforations only. The results of the analysis are evaluated in terms of the parameters."

- Strength of one perforated member is reduced by 17% when compared to non-perforated angle member.
- Strength of two perforated member is reduced by 25.7% when compared to non-perforated angle member.
- trength of three perforated member is reduced by 31.4% when compared to non-perforated angle member.

REFERENCES

1. "Buckling resistance of perforated steel angle members", Journal of Constructional Steel Research, Italy.
2. "Design for openings in equal-angle cold-formed steel stub columns", Structural Steel Research Group, Singapur.
3. "Elastic buckling analysis of thin-walled structural members with rectangular holes using generalized beam theory", Department of Civil and Environmental Engineering, Virginia Tech, USA.
4. "A finite element analysis model for the behaviour of cold-formed steel members", Department of Civil Engineering, McMaster University, Hamilton, Ontario, Canada.
5. "Distortional buckling of perforated cold-formed steel channel section beams with circular holes in web", International Journal of Mechanical Sciences, China.
6. "Behaviour of cold-formed steel perforation sections in compression", Department of steel structures and structural mechanics, "Politechnica" University of Timisoara, Romania.

