IOSR Journal of Engineering (IOSRJEN) **www.iosrjen.org**
ISSN (e): 2250-3021, ISSN (p): 2278-8719
Vol. 08, Issue 12, December 2018, ||Ver. -VI|| PP 01-11

Study On Analysis Of Cold Formed Steel Infilled Concrete Frame Section Using ANSYS Software.

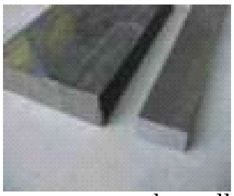
Mrs.Sunitha.K[1], P.Karthigaiselvi[2], K. Muralikrishnan[3]
[3]Assistant Professor, Dept. of Civil Engineering, CMS college of Engineering & Technology, Coimbatore, Tamil Nadu,

Abstract- *Cold-formed steel sections are used joists, Barriers and floor decking in construction industry for past several decades. The development of the cold formed steel hollow sections is being used in beams, columns and truss members of residential buildings and industrial buildings. Generally a hollow section fails due to torsional buckling, local buckling and distortional buckling. In order to overcome this failure in-filled material is used to avoid those buckling. This will have limiting width- to- thickness ratio or diameter –to-thickness ratio. In the present study in-filled material is used as light weight concrete. A non-linear Finite Element modeling is done to analyze cold-formed steel in-filled frame section for residential building under static loading using ANSYS14.5.WORKBENCH. Deflection characteristics, stresses and strains are analyzed for the hollow section with and without infilled concrete.*

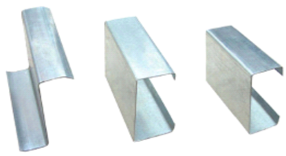
I. INTRODUCTION

In steel construction basically two types of structural steel members are used. There are hot-rolled steel and cold formed steel (fig 1.1). Cold formed steel is made by pressing or rolling gauges of sheet steel under room temperature. When steel is formed by press-braking or cold rolled forming, there is a change in the mechanical properties of the material by virtue of the cold working of the metal. When a steel section is cold-formed one made from flat sheet or strip the yield strength, and to a lesser extent the ultimate strength, are increased as a result of this cold working, particularly in the bends of the section.

Cold formed steel members are widely used in bridges, buildings, storage racks, car bodies, railway coaches and agricultural equipments. Cold formed steel is thinner than hot rolled steel.



hot rolled steel



Cold formed steel

Hot Rolled Steel and Cold Formed Steel

Some of the main properties of cold formed steel are as follows

- Lightness in weight
- High strength and stiffness
- Ease of prefabrication and mass production
- Fast and easy erection and installation
- Substantial elimination of delays due to weather
- More accurate detailing
- Non shrinking and non creeping at ambient temperatures
- No formwork needed
- Termite-proof and rot proof
- Uniform quality
- Economy in transportation and handling
- Non combustibility
- Recyclable material
- Panels and decks can provide enclosed cells for conduits.

CONCRETE FILLED TUBES

Concrete-filled Steel Tube (CFST) type of construction is becoming more popular in today's construction practice. It is a special case of composite column where concrete is filled in a steel tube with or without reinforcement. The steel and concrete combination provides an ideal contribution of strength, whereby the concrete is efficient in compression and the steel in tension. Steel also provides rapid erection and lightweight construction, whilst the concrete is the most economical material when used in compression. Concrete filled steel tube (CFST) has experienced a renaissance in their use in many developed countries such as Australia and Japan.

The CFT column system has many advantages compared with ordinary steel or reinforced concrete systems. The main advantages are listed below:

SCOPE AND OBJECTIVE OF THE WORK

1 A frame for residential building with G+1 floors and single bay is considered for the analysis.

2 Using this frame deflection and strength characteristics of the hollow cold formed steel infilled with concrete is found out using ANSYS software.

3 And it is then compared with hollow cold formed steel without infilling of same frame.

II. LITERATURE REVIEW

Rajeshkumar.B, Anil Kumar Patidar and Helen Santhi.M presented the analytical study on Concrete Filled Cold Formed Steel sections (CFCFS) using ANSYS software. The behaviour of CFCFS sections as beams and columns is studied. The sections consist of a rectangular cross section of size 160 mm x 140 mm with different thicknesses, namely, 3 mm, 4mm, 5 mm and 6 mm. The CFCFS are filled with M20, M30 and M40 grade concrete and the behaviour is compared with the hollow section.

P. Venkata Sumanthchowdary, Senthil Pandian. M studied resistance of cold formed steel in filled frame under static loading. In the study in-filled material is used as light weight concrete. A non-linear Finite Element modeling is done to analyze cold-formed steel in-filled frame section for residential building under static loading using ANSYS12.WORKBENCH. Analysis of the frame gave increasing moment at lower dimensions. Analysis had done for deformation of frame by increasing load.

Nastaran Hosseinjani, Hossein Parastesh presented paper tends to deal with new method of lightening in cold–formed structures group in which inside of empty frames in cold– formed, steel–made buildings have been filled with foamed concrete and in which members connections are designed jointly making use of bolts and nuts. The results of the study have been calculated using modeling in ABAQUS software, in the form of place force–deformation figure.

Ben young and Ehab Ellobody presented an experimental investigation of concrete-filled cold-formed high strength stainless steel tube columns. The behavior of the columns was investigated using different concrete cylinder strengths varied from 40 to 80 Mpa. A series of tests was performed to investigate the effects of the shape of the stainless steel tube, plate thickness and concrete strength on the behavior and strength of concrete-filled high strength stainless steel tube columns. The high strength stainless steel tubes were cold-rolled into square and rectangular hollow sections. The concrete-filled high strength stainless steel tube specimens were subjected to uniform axial compression. The column strength, load-axial strain relationships and failure modes of column were studded.

Alireza Bagheri Sabbagh, Mihail petkovski, Kyprospilakoutas and Rasoul mirghaderi presented Finite Element modeling is used for simulating hysteretic moment-rotation behavior and failure deformation of bolted cold formed steel (CFS) moment connections, tested on a series of six beam column assemblies comprising CFS curved flanged beams, a support column through a plate were tested under cyclic loading. The moment rotation behavior of the connections was dominated either by flexural in the beams or by bolt slip. This paper concludes the modification factor for slip resistance under cyclic loading.

OVERVIEW OF THE STUDY

1 Most of the research effort has taken to study the deflection of cold formed frame.

2 A limited research has been carried out to study about its strength and stress characteristics.

3 And only few research efforts have been taken to compare the cold formed steel with and without infillings.

4 Hence in this study comparison of cold formed steel with and without infillings is to be carried out in all aspect such as deflection, stress and strength.

A frame i.e. model is to be created using ANSYS software. Since the cold form steel fails due to buckling in fill should be given. Similarly another frame without infilling is modeled. After that the domain (frame) is divided into elements. And then boundary conditions and loads should be applied after assigning material properties. Finally the comparison is made between them using the result obtained.

III. ANALYTICAL METHODS

Stiffness matrix method

The direct **stiffness method** was developed specifically to effectively and easily implement into computer software to evaluate complicated structures that contain a large number of elements. Today, nearly every finite element solver available is based on the direct stiffness method. While each program utilizes the same process, many have been streamlined to reduce computation time and reduce the required memory. In order to achieve this, shortcuts have been developed.

One of the largest areas to utilize the direct stiffness method is the field of structural analysis where this method has been incorporated into modeling software. The software allows users to model a structure and, after the user defines the material properties of the elements, the program automatically generates element and global stiffness relationships. When various loading conditions are applied the software evaluates the structure and generates the deflections for the user.

Flexibility matrix method

In structural engineering, the flexibility method, also called the method of consistent deformations, is the traditional method for computing member forces and displacements in structural systems. Its modern version formulated in terms of the members' flexibility matrices also has the name the matrix force method due to its use of member forces as the primary unknowns.

Finite element analysis

This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications. The introduction of FEM has substantially decreased the time to take products from concept to the production line. It is primarily through improved initial prototype designs using FEM that testing and development have been accelerated. In summary, benefits of FEM include increased accuracy, enhanced design and better insight into critical design parameters, virtual prototyping, fewer hardware prototypes, a faster and less expensive design cycle, increased productivity, and increased revenue.

A typical work out of the method involves (1) dividing the domain of the problem into a collection of sub domains, with each sub domain represented by a set of element equations to the original problem, followed by (2) systematically recombining all sets of element equations into a global system of equations for the final calculation. The global system of equations has known solution techniques, and can be calculated from the initial values of the original problem to obtain a numerical answer.

ANALYTICAL SOFTWARES

GENERAL

The computer has become an invaluable aid to the structural engineer. It permits better analysis in much less time than hand methods. It provides the engineer more flexibility to change member sizes and investigate different support conditions, various loading condition and various modeling assumption, than possible with time consuming hand analyses. Use of this greater analysis power removes the tedium of hand analysis and allows much more flexibility, but demands that the responsible engineer become familiar with each program, its capabilities and limitations, and verifies the results of each analysis. However the computer cannot substitute for an engineer's education, experience, judgment and responsibility.

TYPES OF SOFTWARE USED

S-frame software formerly SOFTEK Services Ltd. is a Canadian engineering software company that develops analysis and design software for use by civil and structural engineers especially for plane frame.

SAP 2000 was used to generate 3d FEA models. This computer program was used to generate nodes, elements and 3d meshes.

NASTRAN is primarily a solver for finite element analysis. It does not have functionality that allows for graphically building a model or meshing. All input and output to the program is in the form of text files. However, multiple software vendors market pre- and post-processors designed to simplify building a finite element model and analyzing the results.

ANSYS Structural software

ANSYS addresses the unique requirements of pure structural analysis without the need for extraneous tools. The product delivers all the power of nonlinear structural capabilities as well as all linear capabilities to deliver high-quality, reliable structural simulation results.

Existing native CAD geometry can be used directly with ANSYS structural analysis software — with no translation, no IGES and no intermediate geometry formats. ANSYS has provided native bidirectional integration with the most popular CAD systems for more than a decade. Integration directly into the CAD menu bar makes it very simple to launch world-class simulation directly from a CAD system. Since the ANSYS geometry import mechanism is common to all CAD systems; the user has the flexibility to work within a single common simulation environment while using multiple CAD packages.

In addition, the ANSYS Workbench environment supports neutral format files such as IGES, Parasolid®, ACIS® (SAT) and STEP, which enable the use of any CAD system with the capability to export to any of these formats.

ANSYS provides a wide range of highly robust automated meshing tools — from tetrahedral meshes to pure hexahedral meshes, inflation layers and high-quality shell meshes. Mesh settings like surface or edge sizing, sphere of influence, defeaturing tolerances and more can be set by the user.

Once the geometry has been imported, ANSYS structural software tools automatically detect and perform setup for contacts or joints between parts of an assembly. The contact settings and options can be modified, and additional manual contact definitions can be added. Joints for flexible/rigid dynamics are automatically detected. Each contact or joint is easily identified using the graphical tools provided in the environment.

The current generation of ANSYS element technologies provides rich functionality with a consistent theoretical foundation coupled with the most advanced algorithms. ANSYS structural analysis software provides a large library of elements including beam, pipes, shells, solids, 2-D planar/axisymmetric and 3-D axisymmetric elements, which have wide applicability that includes composites, buckling and collapse analysis, dynamics analysis and nonlinear applications. The library also includes special-purpose elements like gaskets, joints, interface elements and layered elements for composites structures.

These elements offer superior performance and functionality. They also support advanced material models and methods like remeshing/rezoning, fracture mechanics and coupled fields while also accommodating distributed solver processing needs

It is vital to understand and accurately characterize material behavior while designing or analyzing an engineering application. ANSYS provides a vast library of mathematical material models that aid users in simulating various kinds of material behavior, such as elasticity, visco-elasticity, plasticity, viscoplasticity, cast iron plasticity, creep, hyperelasticity, gaskets and anisotropy. These constitutive models can be used to simulate various kinds of materials: metals, rubber, plastics, glass, foam, concrete, bio-tissues and special alloys. In addition, to aid in finding parameters for these materials models, ANSYS provides a set of curve-fitting tools.

The virtual crack closure technique (VCCT) allows computation of energy-release rates for two-dimensional continuum and 3-D continuum elements. Two-D elements also support crack growth simulation.

With a solid foundation of element and material technology, ANSYS structural analysis software offers various advanced modeling methods for different kinds of applications

In addition, ANSYS finite element analysis (FEA) tools offer advanced capabilities that enable simulation of a variety of physics phenomena, such as thermal–stress, electromechanical, structural–acoustics, mass diffusion and simple thermal–fluid analysis.

ANSYS structural analysis software offers a large library of out-of-the box equation solvers. The library contains the sparse direct solver, preconditioned conjugate gradient (PCG) iterative solver, Jacobi conjugate gradient (JCG) solution and more. In addition, the distributed versions of PCG, JCG, and sparse solvers are available for use in large-scale computing via parallel processing. By combining our parallel algorithms with the power of GPUs, you can further reduce the solution time required for large models.

Variational technology from ANSYS allows accelerating computation of normal modes for cyclic structures, especially when a large number of harmonic indexes are required. Frequency sweeps such as those found in harmonic analyses benefit from variational technology as well. Typical speedup factors range from three to 10 times. Transient thermal runs and certain classes of nonlinear structural transient problems are computed in a shorter time using these same principles.

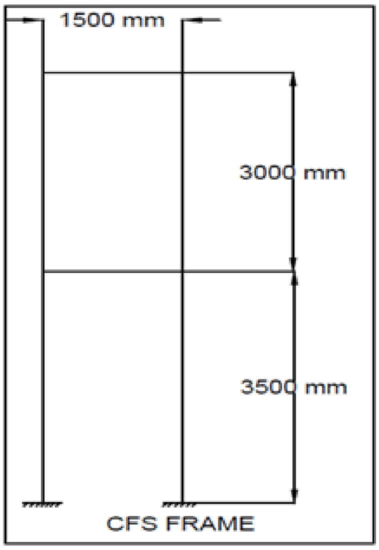
Customization capabilities through user elements, user materials and scripting using ANSYS Parametric Design Language (APDL) provide flexibility and extend the capability of applications for structural analysis solutions.

APDL is the foundation for accessing sophisticated features of the structural solver. In addition, engineers can use APDL to automate common tasks, build their own parametric models, perform design optimization, construct adaptive meshing, etc., as it offers many convenient features such as parameters, macros, branching, looping, and repeating and array parameters that can be used in day-to-day analyses.

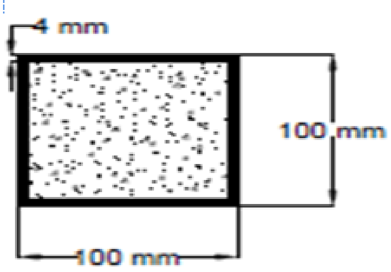
ANSYS FEA software provides a comprehensive set of post-processing tools to display results on models as contours or vector plots to provide summaries of the results (like min/max values and locations). Powerful and intuitive slicing techniques allow the user to get more detailed results over given parts of geometries. All results can be exported as text data or to a spreadsheet for further calculations. Animations are provided for static cases as well as for nonlinear or transient histories. Any result or boundary condition can be used to create customized charts.

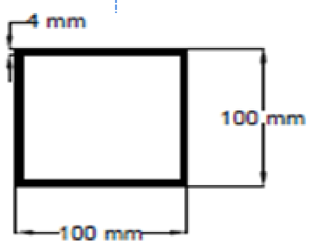
PRE ANALYSIS
MODEL CREATION

A model is created a frame for residential building by using AUTO CAD software. The frame consist G+ 1 floor. Each floor height is 3m and 3.5 meters clear span between two columns while thickness of cold-formed steel used is 4mm.Dimensions of the sections are 100 mm breadth, 100mm depth for both beam and column.. These hollow sections are in filled with solid section of normal concrete. The frame is imported to ANSYS for analysis.



Model in Autocad





CFS With Infilled Concrete	CFS Without Infilled Concrete

PROPERTIES OF MATERIAL USED
Cold Formed Steel

Density	:	7849.05Kg/m
Unit Weight	:	250 MPa
Poisson's Ratio	:	0.3
Elastic Modulus	:	203395 MPa
Shear Modulus	:	76923 MPa
Tangent Modulus	:	1450 MPa

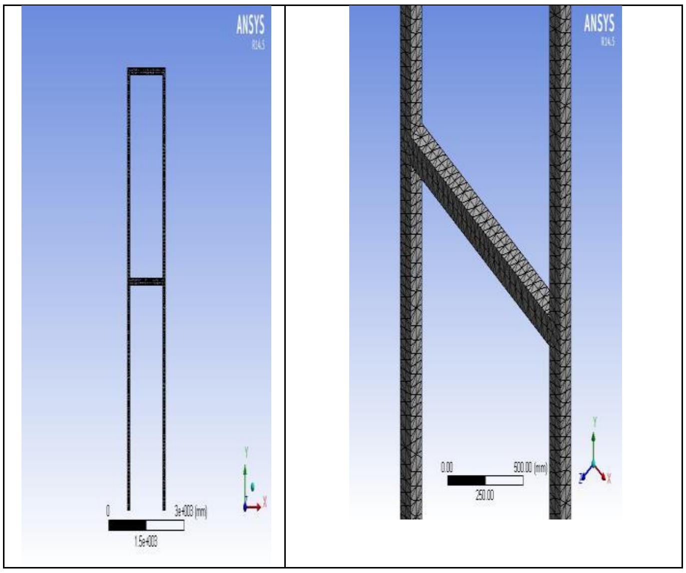
Bulk Modulus	:	1.667×10^{5} MPa

CONCRETE

Density	:	2300 Kg/m^{3}
Poisson's Ratio	:	0.18
Elastic Modulus	:	30000 MPa
Shear Modulus	:	12712 MPa
Bulk Modulus	:	15625 MPa

MESHING

Division of the domain into elements is called mesh. To obtain good results from the solid model is divided into small elements, and the use of tetrahedral mapped mesh is recommended for fem analysis. Therefore, mesh was setup such that tetrahedral elements were created. The meshing is done with mesh tool menu which has global set containing the size of the element divisions which defines the size of the element which is formed. As the size of the elements decreases the elements are increased in number which the results obtained are too accurate. As the elemental number increases the time consuming for solving a problem for the particular load increases thereby requires more memory space in the computer. Each element is assumed to be connected to the neighboring elements only at finite number of discrete points called nodes.

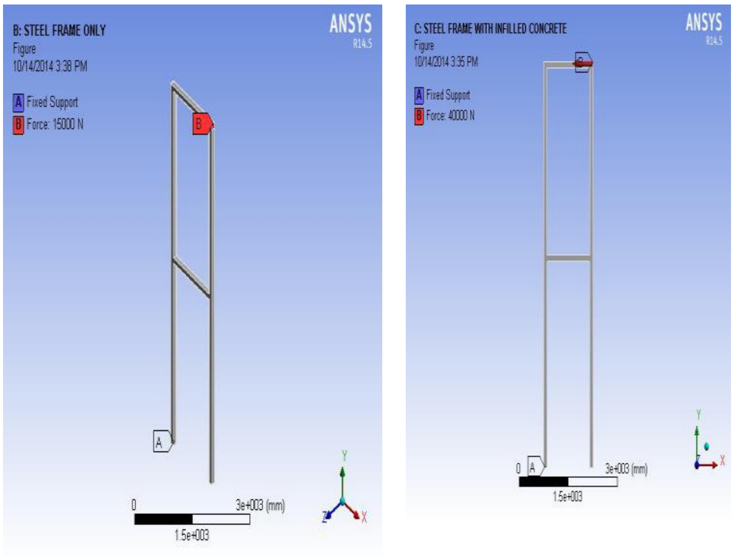


Finite Element Meshing

LOADS AND BOUNDARY CONDITIONS

A building has to perform many functions satisfactorily. Design of building should be structurally safe from loads being acted upon. For the frame with and without infilled concrete a lateral load of 40 kN and 15 kN is given after many trails.

Displacement boundaries are needed to constraint the model to get a unique solution. To ensure that the model acts in the same way as experimental from boundary conditions, which are applied at the points of symmetry .The support was modeled as fixed support.

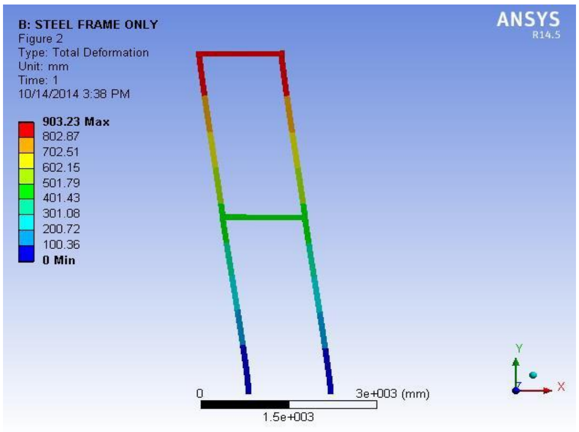


Application of boundary conditions and loads to the frame

POST ANALYSIS
HOLLOW CFS FRAME
Total Deformation

Deformation refers to any changes in the shape or size of an object due to an applied force (the deformation energy in this case is transferred through work) or a change in temperature (the deformation energy in this case is transferred through heat).

Maximum and minimum deformation obtained for the hollow CFS frame is 903.23mm and 100.36mm respectively as shown in figure below.

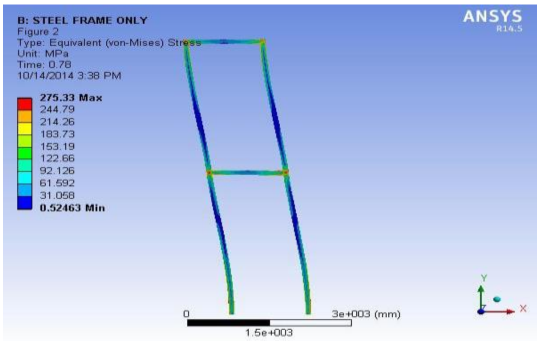


Total Deformation of Hollow CFS Frame

Von Mises Stress

Von Mises stress is a geometrical combination of all the stresses (normal stress in the three directions, and all three shear stresses) acting at a particular location.

Maximum and minimum Von Mises (equivalent stress) obtained for the hollow CFS frame is 275.33 mm and 0.52463 mm respectively as shown in figure below

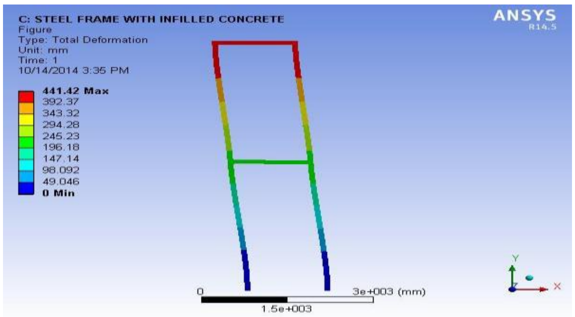


Equivalent Stress of Hollow CFS Frame

CONCRETE FILLED CFS FRAME

Total Deformation

Maximum and minimum deformation obtained for the concrete filled CFS frame is 441.42 mm and 49.046 mm respectively as shown in figure below.

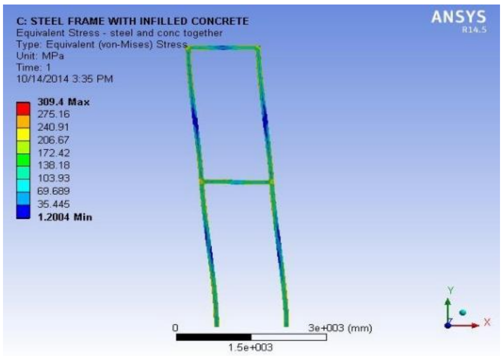


Von Mises Stress

Total Deformation of Concrete Filled CFS Frame

Concrete alone

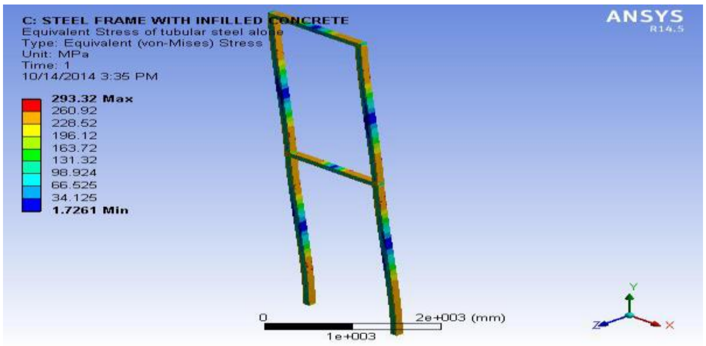
Maximum and minimum Von Mises (equivalent stress) obtained for the frame is 309.4 MPa and 1.2004 MPa respectively as shown in figure below .



Equivalent Stress of Concrete Filled CFS Frame (steel and concrete)

Steel frame alone

Maximum and minimum Von Mises (equivalent stress) obtained for the hollow CFS frame is 293.32 MPa and 1.7261MPa respectively as shown in figure below.



Equivalent Stress of Concrete Filled CFS Frame (hollow steel alone)

THEORETICAL INVESTIGATION

GENERAL

For validating an analysis theoretical or experimental work is essential. In order to validate the result obtained from analysis slope deflection method is used to find the deflection at the joints. The slope deflection method is a structural analysis method for beams and frames introduced in 1914 by George A. Maney. The slope deflection method was widely used for more than a decade.

MAXIMUM DEFLECTION

1 Frame without infill = 0.33861 + 0.55064

= 889.22mm

2 Frame without infill = 0.186 +0.25405

= 440.05mm

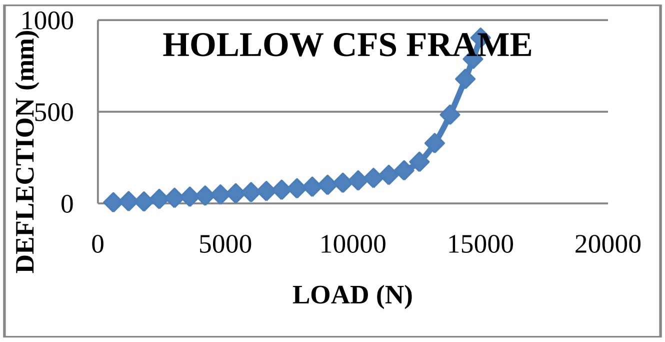
The difference between the theoretical and analytical method is about 2-3%

RESULTS AND DISCUSSION

Ffrom the value obtained from the analysis a curve is drawn between deflection and load for the both cases as follows.

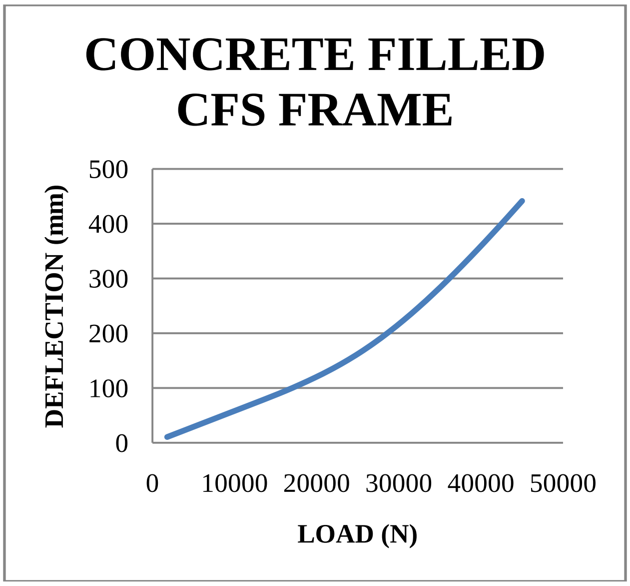
Load N	DEFLECTION mm	Load N	DEFLECTION mm
600	6.0904	8400	90.992
1200	12.181	9000	101.2
1800	10.449	9600	112.63
2400	24.362	10200	125.11
3000	30.452	10800	138.86
3600	36.543	11400	155.6
4200	42.634	12000	180.11
4800	48.74	12600	227.02
5400	54.9	13200	328.71
6000	61.186	13800	483.58
6600	67.667	14400	678.46
7200	74.511	14700	787.28

CONCRETE FILLED CFS FRAME



Load Vs Deflection Curve for CFS without Infilled Concrete

Load N	Deflection mm	Load N	Deflection mm
1800	10.449	25200	164.01
3600	20.9	27000	182.29
5400	31.35	28800	202.31
7200	41.8	30600	224
9000	52.25	32400	247.24
10800	62.7	34200	271.92
12600	73.172	36000	297.89
14400	83.722	37800	324.97
16200	94.672	39600	352.99
18000	106.45	41400	381.81
19800	118.98	43200	411.31
21600	132.55	45000	441.42
23400	147.52		



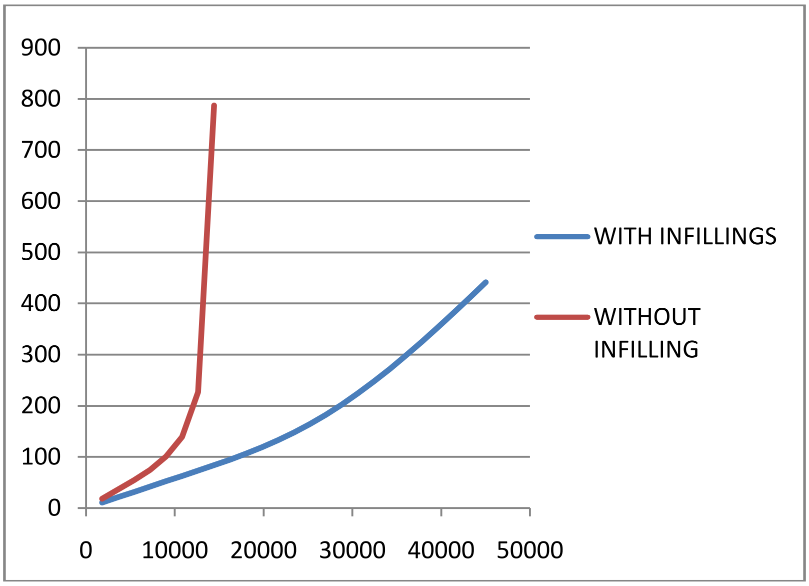
Load Vs Deflection Curve for CFS with Infilled Concrete

COMPARSION

Comparison is made between the CFS infilled concrete and CFS without infilled concrete. From that it is studied that CFS infilled concrete takes more load compared to Load Vs Deflection Curve for CFS without Infilled Concrete other.

Load N	Deflection in mm	
	With infilling	Without infillings
1800	10.449	18.271
3600	20.9	36.543
5400	31.35	54.9
7200	41.8	74.511
9000	52.25	101.2
10800	62.7	138.86
12600	73.172	227.02
14400	83.722	787.28

Comparison of CFS with and without infilled concrete



Comparison of CFS with and without infilled concrete

IV. CONCLUSION

This paper presents the non-linear finite element analysis of cold formed steel frame with and without infilled concrete under static loading. The following conclusions are derived based on the analysis.

1. The frame without infillings is getting maximum allowable deflection under lateral load of 15 kN.
2. Whereas the infilled frame undergoes maximum allowable deflection under lateral load of 45kN.
3. Concrete –in filling in hollow sections decrease the width- to-thickens ratio and improve the load carrying capacity of the frame.
4. The proposed cold-formed steel in filled frame shows better resistance towards the deformation.
5. Large deformation of static loading reduced due to using in fill material in the hollow sections.
6. Difference between the theoretical and analytical method is about 2 to 3%

REFERENCES

[1]. nptel.ac.in/courses/105106113/5_cold_form_steel/1_introduction.pdf
[2]. www.webcivil.com/uscfproperty.aspx
[3]. en.wikipedia.org/wiki/Ansys
[4]. en.wikipedia.org/wiki/Cold-formed_steel
[5]. http://www.ansys.com/Products/Simulation+Technology/Structural+Analysis/ANSYS+Structural
[6]. Resistance of cold formed steel in filled frame under static loading by P. Venkata Sumanthchowdary, Senthil Pandian. M
[7]. FE analysis of concrete filled cold formed steel sections using ANSYS by Rajeshkumar.B, Anil Kumar Patidar and Helen Santhi. M
[8]. nptel.ac.in/courses/105105109/pdf/m3l22.pdf