IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

INVESTIGATION OF BEHAVIOUR OF COLD FORMED STEEL MEMBERS

[1]N.Boopalraj,[2]M.Vinoth,[3]M.Ramavel, [4]C.Franklin

[1]Assistant professor,[2]Assistant professor, [3]Assistant professor, [4]Assistant professor

[1]Civil Engineering, [2]Aeronautical Engineering, [3]Mechanical Engineering, [4]Mechanical Engineering, J J College of Engineering and technology,Tiruchirappalli,India

Abstract: Thin-walled cold-formed steel (CFS) members are composed of stiffened, partially stiffened, or unstiffened elements. Cold-formed sections have widespread use in the building industries. They are used where the loads are not large and thus where light members will suffice, achieving substantial economics. In addition, they are easily fabricated into a great variety of shapes, and may be formed in sheet metal shops which are widely available. Cold-formed thin-walled steel beams may undergo buckling modes such as local buckling, distortional buckling lateral-torsional buckling or a combination of these before failure. It is necessary to investigate the behavior of cold formed steel members under buckling. The buckling behavior of various available sections and also that of new sections can be analyzed using softwares and compared with experimental results. Direct strength method also can be studied to calculate the capacity of CFS sections, and also to compare the accuracy. The study of CFS beams can be extended to CFS buildings, which also has better lateral load carrying capacity.

Index Terms - **Thin-walled cold-formed steel, CFS beams, buckling behaviour, ABAQUS, Direct strength method (DSM).**

I. INTRODUCTION

Cold-formed sections have widespread use in the building industries and transportation vehicles. They are used where the loads are not large and thus where light members will suffice, achieving substantial economics. In addition, they are easily fabricated into a great variety of shapes, and may be formed in sheet metal shops which are widely available. Cold-forming are made from flat sheet of strip, either by brake forming or by roll forming. In contrast to hot-rolled members who are subjected to residual cooling stresses, cold- formed sections are subjected to strain hardening caused by cold-forming, which may significantly affect the structural performance. Even more important, while the width to thickness ratios of the component plate elements is limited in hot-rolled sections by the manufacturing processes, the cold-forming process allows for practical unlimited width-thickness ratios. With large width- thickness ratios, local plate buckling can occur prior to attainment of the maximum member strength. Both the buckling behavior and the properties of CFCs are different from those of hot-rolled sections. Cold-formed steel members are an economic solution for many construction applications in buildings. The sections are cold-formed from steel sheets, strips, plates or flat bars by using roll-forming, press brake, or bending machines. Cold formed steel members have several advantages over other materials. Cold Formed thin steel sheet products are extensively used in building industry, and range from purlins to roof sheeting and floor decking. Generally these are available for use as basic building elements for assembly at site or as prefabricated frames or panels. These thin steel sections are cold-formed, that is their manufacturing process involves forming steel sections in a cold state from steel sheets of uniform thickness. These are called as Cold Formed Steel Sections. Sometimes they are also called light gauge steel sections or cold rolled steel sections. The thickness of steel sheet used in cold formed construction is usually 1 to 3mm. Much thicker material up to 8 mm can be formed if pre galvanised material is not required for the particular application. The method of manufacturing is important as it differentiates these products from hot rolled steel sections. Cold- formed steel (CFS) is the common term for steel products shaped by cold-working processes carried out near room temperature, such as rolling, pressing, stamping, bending etc. Thin-walled cold-formed steel (CFS) members are composed of stiffened, partially stiffened, or unstiffened elements. Cold-formed thin-walled steel beams may undergo buckling modes such as local buckling, distortional buckling lateral-torsional buckling or a combination of these before failure as shown in figure 1.3. Failure may be due to

one of the modes or interaction of the different buckling modes. The thin member's elements (flange, web, lip, etc) may buckle locally before reaching yield stresses when the member is subject to compressive, flexural, shear, or bearing loads. Local buckling is a major consideration in design of cold-formed steel members; this design must provide a good safety margin against local instabilities. The strength of thin-walled cold formed steel (CFS) compression members may be governed by yielding (Y), local (L), distortional (D), or global/Euler's (E) buckling. The failure may also be due to any possible interaction among these buckling modes, such as local-distortional (LD), local-global (LE), distortional-global (DE), or local-distortional-global (LDE). Such interactions reduce the strength of the members to values below the strength as affected by any of the independent buckling modes. Hence, any design method should accurately evaluate not only the strength of members as affected by the three different independent buckling modes, but also the strength considering the possible interaction of the buckling modes. The direct strength method (DSM), which is explained in section 1.6 has been introduced as a simpler and more accurate method for evaluating the strength of prequalified CFS members.

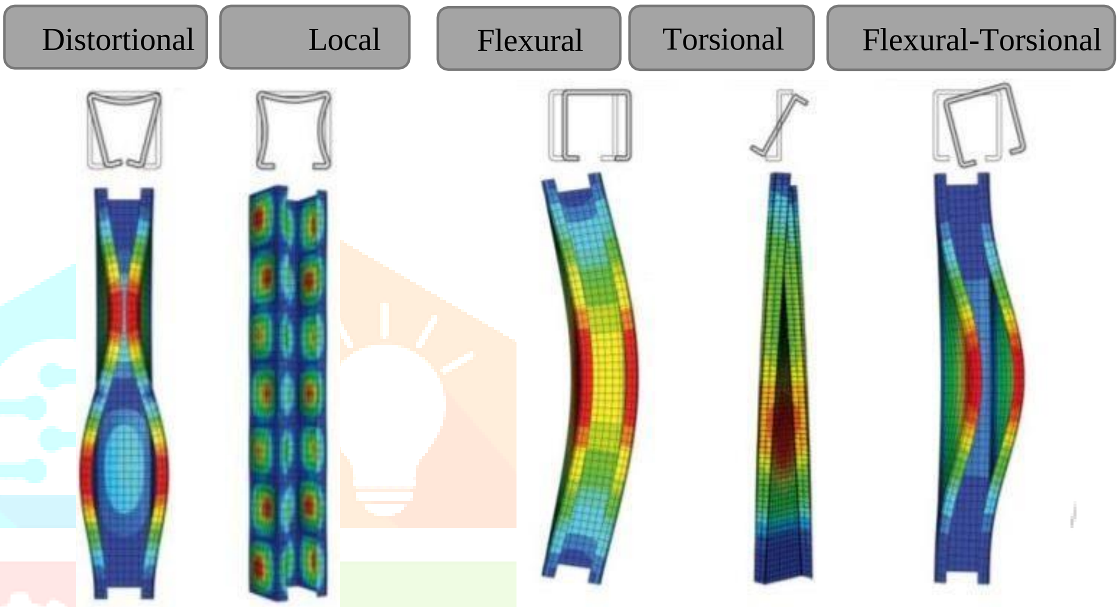


Figure 1. Failure modes of CFS members

II. THEORETICAL STUDY

The Effective Width Method (EWM) and the Direct Strength Method (DSM) are two major approaches considered in design specifications of CFS members. EWM was initially introduced by Von Karman et al (1932) and was later modified by Winter (1947) for CFS. It has now been well established and adopted in design specifications such as Eurocodes (EN1993-1-3, 2006), American Iron and Steel Institute (AISI, 2007) and Australian/New Zealand Standard (AS/NZS, 2005). DSM a relatively recent concept was developed by Schafer and Peköz (1998a), has been included as an alternative procedure in American Iron and Steel Institute in 2004 (AISI, 2007) and Australian/New Zealand Standard for CFS strength determination.

Effective Width Method (EWM)

The basic idea of the EWM is that local pate buckling leads to reductions in the effectiveness of the plates that comprise a cross-section in isolation; furthermore, it can be better understood as an idealized version of equilibrium in an effective plate under a simplified stress distribution vs. the actual plate with a nonlinear stress distribution due to buckling. For a uniformly compressed rectangular plate the stress distribution is uniform prior to buckling load. As the plate starts to buckle, it deflects laterally and experiences a stress gradient, with the central area unable to support any additional load although portions close to the supported edges continue to carry increasing load. This stress distribution for the post-buckling stage is complicated in nature and difficult to calculate in practice, since the variation for individual members can be significantly different. Instead, this loss in plate effectiveness can be considered as an approximate means to represent the equilibrium of stress applied on the partial width of the plate, particularly as ' the effective width'.In the design codes integrated EWM includes effective width for both web and compression flanges when the section is considered under local buckling. In Eurocode 3, the theory of EWM extended to a 'reduced thickness' of edge or intermediate stiffeners for the effect of distortional buckling. However, inter- elements i.e. between the flange and the web, are not taken into account since EWM considers each element separately hence this method is not accurate for complex sections. Moreover, with the development of complex shapes such as additional folds and stiffeners added to the sections, determining the effective width of the section becomes increasingly more complicated.

Direct Strength Method (DSM)

In the meantime, an alternative method developed by examining the elastic buckling solutions for the whole cross-section rather than the individual elements, and strength curves for the entire member called Direct Strength Method (DSM). DSM assumes that local buckling behaviour can also be predicted by the elastic buckling stress of the entire section with a suitable strength design curve for local instability. In DSM, all of the elastic instabilities for the gross section i.e. local (Mcrl), distortional (Mcrd), and global buckling (Mcre) are determined and also the moment (or load) that causes the section to yield (My) is determined. Therefore, the strength (Mn) can be directly determined by Mn = f (Mcr`, Mcrd, Mcre, My). Moreover, DSM delivers an equivalent level of accuracy and simple calculation procedure in comparison with EWM for predicting CFS member capacity, particularly for complicated cross-section geometry, since it does not require calculating the stress for each individual element within the member. Both EWM and DSM provide a simplified solution to an initially complex nonlinear problem with CFS member to allow engineers have a working model to design without the requirement of testing every individual member.

III. NUMERICAL INVESTIGATION

General:

This section includes a modeling and validation study of a simply supported CFS Channel beam subjected to transverse loading at mid-length. The buckling analysis both linear and non-linear was done on the model created in ABAQUS software. The results were compared with the journal (Nandhini et.al.(2010)) to validate the study.

Finite Element Method (FEM):

Advances in the field of computer-aided engineering have led the use of finite element analysis (FEA) in thin-walled steel structures increasing rapidly in the last decade due to its higher accuracy for analysing numerically. In FEM, the domain is divided into smaller discrete sections called finite elements, which are interconnected at nodal points resulting in set of simultaneous algebraic equations. FEM then solve these equations using partial differential equations by replacing continuous functions with piecewise approximations defined on polygons. The distribution of elements within the domain called mesh is geometry dependent as well as on the precision of the solution. Therefore, FEM solves by reducing the problem into a set of linear equation using polynomial approximations. Displacement based finite element method is usually used in structural analysis, in which a set of functions are established such that they distinctively define the state of the displacement within each element in terms of its nodal values. The number of degrees of freedom is now finite and displacement for each element is approximated using shape function and the nodal values. Hence, displacement at each nodal point is calculated by solving these equilibrium equations. Generally, FEA is consists of three fundamental steps:

1. Pre-processing - At this stage model is developed and its geometry is meshed to divide into finite elements, and material properties, loads, and boundary conditions are applied.
2. Analysis – The dataset prepared at earlier stage is applied to generate matrix equations for each element, which are then assembled and solved numerically for the type of analysis (e.g. static or dynamic) selected. The form of equations is always;

$$K_{ij}u_j = f_i$$

   Where, u and f are the displacement and applied load at the nodal points and the formation of the K matrix is dependent on the type of problem being chosen.
3. Post-processing – The obtained results are validated and examined in terms of displacement and stresses at a discrete position within the model using graphical representation. The finite element based commercial software ANSYS and ABAQUS are commonly used to predict the structural behavior and ultimate strength of CFS structural members.

ABAQUS Software

ABAQUS is general purpose software based on FEM that can be used for micro- element modelling of CFS members. However, ABAQUS can also be used for macro-element modelling when simplification assumptions are made, which is out of the scope of this research

ABAQUS modeling

The finite element model can be analysed using a finite element program such as ABAQUS. During this process, ABAQUS program performs a sequence of statistical tasks using a system of simultaneous equations generated from the model and resultant nodal values are stored. The program offers various types of equation solvers subject to the type of analysis. The type of analysis could be linear or non- linear and is specified at the beginning of the solution process. The structural behaviour of a model could be analysed based on its linear or nonlinear response to external forces.

Material Properties

Material nonlinearity in the CFS members was modelled with von Mises yield criterion and isotropic hardening rule16 . Specifications of materials in the software are usually based on coupon tests. Real stress and strains introduced to ABAQUS software can be obtained by converting nominal stress-strain to real.

Element Type

Several element types such as solid two and three dimensional (2D & 3D) elements, membrane and truss elements, beam elements, and shell elements are used for simulation in ABAQUS. Beam, membrane and truss elements might not be suitable for the buckling problems. Solid 3D elements can be used; however, it needs a fine mesh for modelling of high curvature zones. A finer mesh can increase the time of analysis and also does not necessarily take more degrees of freedom into account. For bucking analysis, the suitable element is identified as shell elements herein. S9R5, S4, and S4R elements are three well-known finite elements in ABAQUS which are used for elastic buckling analysis of thin-walled structures. The S4 and the S4R finite elements employ linear functions for inserting deformation between nodes. These elements are four-node shell elements which can be used for thin and thick members. The S9R5 element is a nine-node doubly- curved thin shell element which uses shear flexible strain definitions and Kirchoff constraints enforced as penalty functions16 . By increasing the number of nodes from 4 to 9 it uses quadratic shape functions that can have some benefits. This element has the ability to define initially curved geometries and to simulate a half sine wave with one element reasonably accurate. The 5 in S9R5 shows this element has five degrees of freedom in which the rotation of node about the axis normal to the element mid-surface is omitted from the element formulation for having better computational efficiency. The suffix “R” also shows that reduced integration is used for calculation of element stiffness which needs sensitivity analysis to be used. In this research, S4R was considered for spatial discretization.

Buckling Analysis

The strength of thin-walled cold formed steel (CFS) compression members may be governed by yielding, local, distortional, or global/Euler's buckling. The failure may also be due to any possible interaction among these buckling modes, such as local-distortional, local-global, distortional-global, or local-distortional- global. Such interactions reduce the strength of the members to values below the strength as affected by any of the independent buckling modes. Linear and Non- linear buckling analysis are done on CFS members to find their load carrying capacity.

Linear Analysis

In order to understand the nonlinear system, it is essential to define linear systems first. A linear system is defined such that input-output the relationship is linear. For instance, in structures, when an applied load is doubled, the displacement will also be doubled. Therefore, the method of superposition can be used to solve the linear system again when applying different magnitude of the load. Furthermore, a structural linear system can be illustrated by the flow of physical quantities within them. When the load is applied to the system, local stresses are produced against the globally applied load. In an elastic system, deformation of shape leads to stresses and consequently strains are developed. Strains are integrated at each point to generate the displacement in the global level. When all relationships between loads, stresses, strains and displacement are linear, then the system is called linear. If any of these physical quantities is not linear, them the structural system becomes nonlinear. Linear analysis delivers a fast solution with less amount of modelling time. However, small displacement and rotations in the linear of the material behaviour will give accurate solutions.

Non-Linear Analysis

In practice, structures show nonlinear behaviour within their actual loading ranges, principally due to changing status including contact, geometric nonlinearities and material nonlinearities. Several common structural features exhibit nonlinear behaviour which is statusdependent. Therefore, status changes are either related directly to load or are determined by other external cause. When the structure is undergoing large deformation, its changing geometric configuration can lead to nonlinearity called as geometric nonlinearity. It is characterized when deformed and undeformed state of the structure is significantly different. Material nonlinearity occurs due to the steel nonlinear stress-strain relationship. Hyperelastic, elastoplastic, or viscoelastic materials are examples of nonlinear materials. ABAQUS uses the "Newton-Raphson" approach to solve nonlinear problems in which the load is subdivided into a series of load increments applied over several load steps. The Newton-Raphson method evaluates the difference between the restoring forces and loads, then using this evaluation program perform a linear solution and checks for convergence. This iterative procedure continues until the convergence criteria is satisfied.

IV. RESULTS AND DISCUSSION

Non-Linear Analysis:

A model with initial imperfections corresponding to different buckling modes was generated for the nonlinear (material and geometric) analysis. The nonlinear static analysis was carried out on the models with initial imperfections using the Newton Raphson method to obtain sub-ultimate behaviour, ultimate failure loads and failure modes of the beam. The ultimate load got after non-linear analysis is compared to the numerical results from journal [1] in table 4.3 and graphically represented in figure 4.6.

Figure 2. Specimen after Non-linear analysis

Table 1. Result from Non-linear analysis

RESULTS OF NON-LINEAR ANALYSISUSING ABAQUS	ULTIMATE LOAD (kN)
From Journal	5.123
From Analysis	5.270

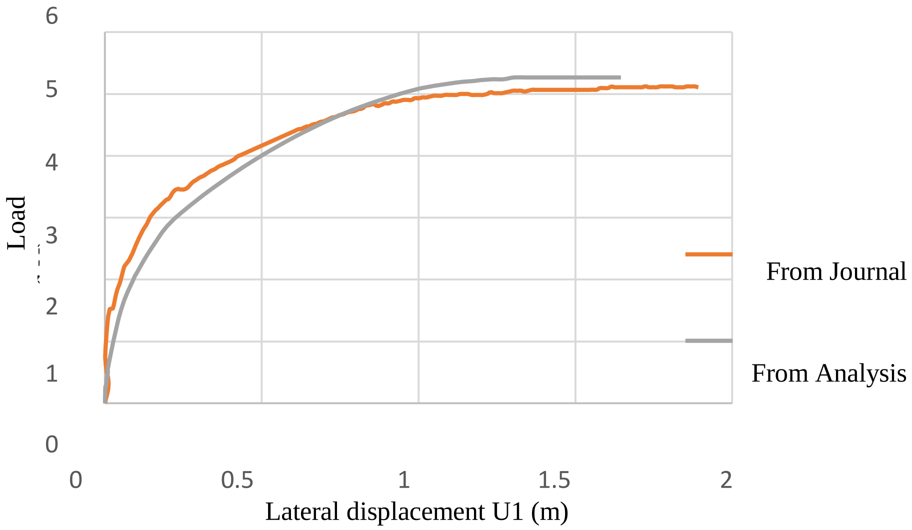


Figure 3. Load-Lateral displacement curve

Buckling analysis on Channel section beam

Modeling: A channel section beams of varying length with the dimensions mentioned in table.2 was modelled in ABAQUS. The material properties used are mentioned in table 2.

Table 2. Dimensions and Material properties of Channel section beam

Webdepth (mm)	Flange width (mm)	hickness (mm)	Youngs Modulus (GPa)	ield Stress(MPa)	oissonsratio
160	70	5	210	355	0.3

Loading and Boundary condition: The beams were modelled with simply supported boundary conditions at the NA node and the lateral displacements at the web-flange junction were restricted at all four corners. Concentrated loading was applied through the shear center at quarter span locations.

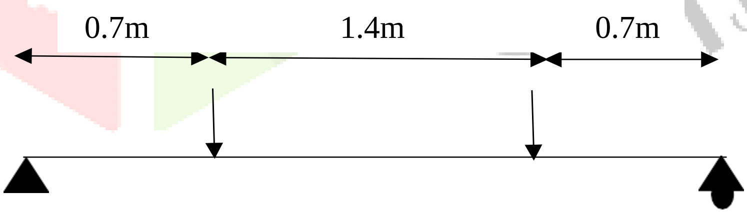


Figure 4. Loading and Boundary condition of 2.8m long beam

Mesh Details: Mesh convergence study was performed in ABAQUS by conducting various analysis by varying the mesh size. The result of mesh convergence study is shown in table 4.5 and also represented in figure 4.7. The results from buckling analysis converges when the mesh sizes are reduced. Similar buckling loads were obtained for mesh sizes less than 15mm. And thus a mesh size of 15mm was chosen for the study. Although further reduction in the mesh showed similar results, the time taken for analysis was more.

Table.3. Mesh Convergence results

Mesh size	No of Elements	Critical Load (kN)
70	444	57.065
50	788	56.137
30	1912	55.773
20	2416	55.125
15	4622	54.787
10	8464	54.715
8	17792	54.672
5	59056	54.572

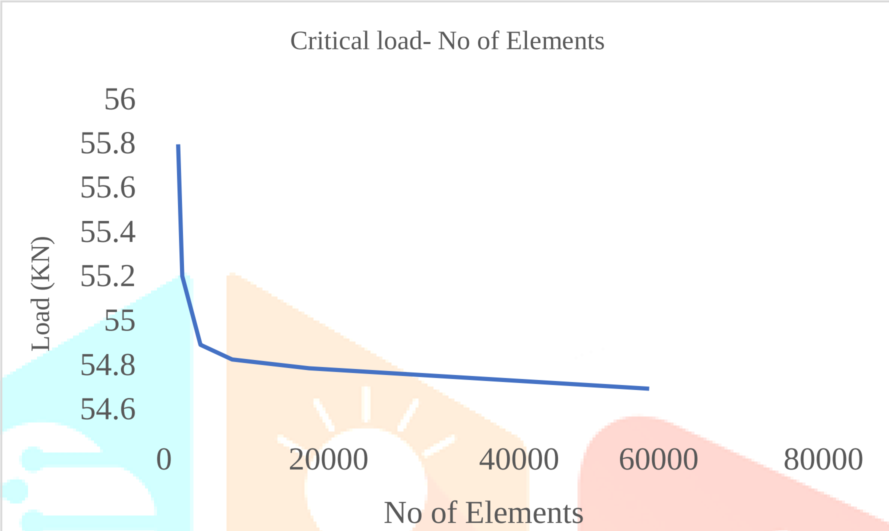
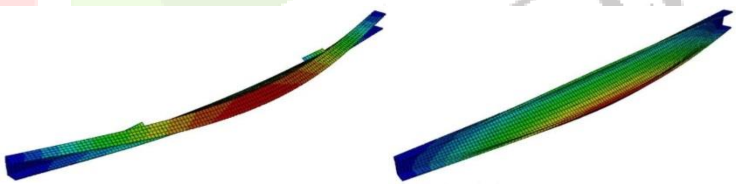


Figure 5. Mesh convergence curve

Analysis and Results:

Buckling analysis was done on the channel beam by varying the length. Linear buckling analysis was conducted to get the critical buckling load and mode shapes of all the models with different length but same cross-section and material properties. The variation of buckling load when lemgth varied is shown in figure 5. This curve shows that as length increases the buckling load decreases. Later, non-linear analysis was conducted on the models to get the ultimate failure loads. The results are given in table 3



Mode 1 Mode 2

Figure 6. Analysis Result

References

[1] P. Nandini and V. Kalyanaraman, 2010, *Strength of cold-formed lipped channel beams under interaction of local, distortional and lateral torsional buckling*, Thin-Walled Structures 48,827-877

[2] Bogdan M. Put,1 Yong-Lin Pi, and N. S. Trahair., 1999, *Lateral Buckling Tests On Cold-formed Channel Beams*, Journal of Structural Engineering, Vol. 125.

[3] K. C. Kalam Aswathy and M. V. Anil Kumar, 2020, *Unstiffened Elements as Limiting Case of Distortional Buckling of Partially Stiffened Elements* , Journalof Structural Engineering, 146(9)

[4] G Beulah Gnana Ananthia, S Vishuvardhanb & G M Samuel Knight, 2015, *Experimental, theoretical and numerical study on thin walled steel single and compound channel sections in axial compression*, Indian Journal of Engineering & Materials Sciences,Vol 22,570-580

[4] Yu. Chen, Xixiang Chen & Chaoyang Wang, 2015, *Experimental and finite element analysis research on cold-formed steel lipped channel beams under web crippling,* Thin-walled structures 87,41-52.