

Parametric Study on Structural Behaviour of Cold Formed Steel Double Channel Hat Section with Two Plates

Rooba N¹, Mrs. D.Amali²

¹PG Scholar, Department of Structural Engineering, Government college of Engineering – Salem-11, Tamilnadu, India

²Assistant Professor, Department of Civil Engineering, Government college of Engineering – Salem-11, Tamilnadu, India

INTRODUCTION

• GENERAL

In steel construction, there are two main families of structural members

- Hot rolled members
- Cold formed members

The use of hot-rolled steel sections become uneconomical for the steel structures subjected to light and moderate loads and for the structural members of short span lengths(e.g., joists, purlins, girts, roof trusses, complete framing of one and two storey residential, commercial and industrial structures).So the study on behaviour of COLD-FORMED STEEL members reduces the cost of a building made up of steel structures.

• ADVANTAGES OF COLD-FORMED STEEL STRUCTURES

Cold forming has the effect of increasing the yield strength of steel, the increase being the consequence of cold working well into the strain-hardening range.

1. Cross-sectional shapes are formed to close tolerances and these can be consistently repeated as long as required.
2. Cold rolling can be employed to produce almost any desired shape to any desired length.
3. All conventional jointing methods, (i.e. riveting, bolting, welding and adhesives) can be employed.
4. High strength to weight ratio is achieved in cold-rolled products.
5. High strength to weight ratio is achieved in cold-rolled products.
6. High strength , stiffness and lightness.
7. Fast and easy erection and installation.

WHAT IS PARAMETRIC STUDY :

- Parametric studies used to nominate parameters for evaluation, define the parameters range, specify the design constraints, and analyze the results of each parameter variation.
- Four different types of cross – section dimensions, four different D/B ratio thickness of the specimen varies are 1,1.25,1.5,1.75 & 2 and three different yield stress (250, 350 & 450 N/mm²) were considered for parametric study by using ABAQUS Software.

BUCKLING:

The sudden change in shape of structural component under load is called buckling. Types of buckling

1. Local buckling
2. Distortional buckling
3. Lateral buckling.

LOCAL BUCKLING:

It is the modes involve the buckling of cross-section walls. this failure mode commonly observed in thin-walled structural steel elements.

DISTORTIONAL BUCKLING:

It is a mode characterized by rotation of the flange at the flange/web junction in members with edge stiffened elements.

LATERAL TORSIONAL BUCKLING:

Buckling that does not involve distortion of the cross-section, instead translation (flexure) and/or rotation (torsion) of the entire cross-section occurs. When a beam fails by lateral torsional buckling, it buckles about its weak axis, even though it is loaded in strong plane. It is also known as global buckling.

Finite Element Method (FEM) :

The finite element method (FEM) is a numerical procedure for analyzing structures and continua. FEM originated in structural engineering, and has been used in the fields of heat transfer, fluid flow, electric and magnetic fields and many others. FEM is the principle applicable to any boundary condition, geometry and material variation. The finite element method has been proven as an efficient and powerful approach to calculate the elastic buckling load and ultimate strength of cold-formed steel (CFS) structural members. A successful static analysis of the unstable collapse and post-buckling behavior of CFS members requires the nonlinear solution method consider geometric nonlinearity, material nonlinearity, boundary nonlinearity and residual stresses of the physical objects, as well as have the capability to deal with convergence, locking and other difficulties related to implementing the numerical algorithm.

FINITE ELEMENT MODELLING

Finite elements models were based on the centre line dimensions of the cross sections with the plate thickness.

Part module

The part module allows you to create individual parts by sketching their geometry directly in abaqus or by importing their geometry from other geometric modeling programs.

Property module

The material properties like young's modulus, Poisson's ratio, yield stress will be defined to the sections.

Assembly module

An assembly is a collection of positioned part instances. An analysis is conducted by defining boundary conditions, constraints, interactions, and a loading history for the assembly.

Part instance

A part instance is a usage of a part within the assembly. All characteristics (such as mesh and section definitions) defined for a part become characteristics for each instance of that part, they are inherited by the part instances. Each part instance is positioned independently within the assembly.

Model Discretization

The numerical models were discretized with the reduced integration four noded shell element with reduced integration was selected from the ABAQUS element library. This element uses three translational and three rotational degrees of freedom at each node (S4R). The mesh size for the shell elements given is equal to the thickness of the section is found to hold good simulation results.

Step module

To create and configure analysis steps and associated output requests.

The step sequence provides a convenient way to capture changes in a model (such as loading and boundary condition changes).

Interaction module

It defines the contact information of the section.

In this module

- Reference points are created at both the ends & loading points.
- Constraints are created for various sections. The most commonly used section with Constraint.

Loading and Boundary condition:

The Boundary condition introduced to the centroid node is shown in Fig 6.4. Transverse load is represented as a concentrated nodal force. It is disturbed to the loading points through coupling constraints. The simply supported end conditions of the beam are modelled as prevented from rotations about the y and z-axis, and translations x, y and z-directions. The load will be applied at the reference point (in the z-axis direction).

Mesh module

It contains tools that allow us to generate a finite element mesh on an assembly created within abaqus.

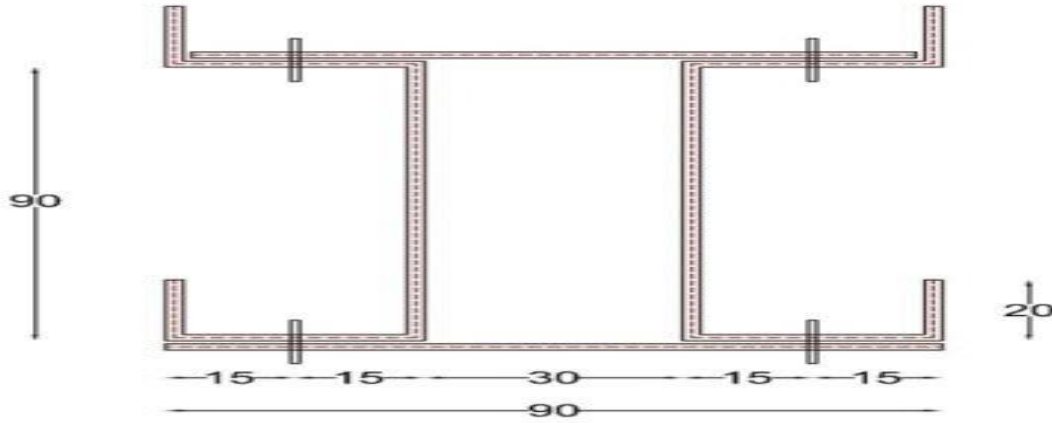
Job module

It is used to analyze the model. It allows us to interactively submit a job for analysis and monitor its progress.

Visualization module

It provides graphical display of finite element models and results.

DC-(90 X 90 X 2) – 2400mm(Model section)



Abaqus results of following sections:

SI.NO	D/B RATIO	SECTION (mm)	Thickness	f_y	Maximum Load KN	M_{FEM}	Mode of failure
			T (mm)	N/mm ²		KN m	
1	1	DC - 90X90	1.6	250	11.098	3.884	L
2				350	13.405	4.691	L
3				450	15.680	5.488	L
4			2	250	14.695	5.143	L
5				350	16.793	5.877	L
6				450	17.695	6.193	L

SI.NO	D/B RATIO	SECTION (mm)	Thickness	f_y	Maximum Load KN	M_{FEM}	Mode of failure
			T (mm)	N/mm ²		KN m	
7	1.25	DC - 90X112.5	1.6	250	20.798	7.279	L
8				350	21.821	7.637	L
9				450	22.080	7.728	L
10			2	250	21.625	7.568	L
11				350	22.031	7.710	L
12				450	23.178	8.112	L

SI.NO	D/B RATIO	SECTION (mm)	Thickness	f_y	Maximum Load KN	M_{FEM} KN m	Mode of failure
			T (mm)	N/mm2			L - (Local Buckling)
19	1.75	DC – 90X157.5	1.6	250	22.281	7.798	L
20				350	24.161	8.456	L
21				450	26.172	9.160	L
22			2	250	24.854	8.698	L
23				350	25.483	8.919	L
24				450	27.511	9.628	L



SI.NO	D/B RATIO	SECTION (mm)	Thickness	f_y	Maximum Load KN	M_{FEM} KN m	Mode of failure
			t (mm)	N/mm2			L - (Local Buckling) D – (Distortional buckling)
25	2	DC - 90X180	1.6	250	24.631	8.620	L
26				350	25.035	8.762	L
27				450	25.864	9.052	L
28			2	250	26.272	9.195	L
29				350	27.458	9.610	L
30				450	28.311	9.908	L



CONCLUSION

In this project, the load carrying capacity and failure modes of built-up I-sections have been analyzed by ABAQUSE Software.

- The result of linear and non-linear numerical analysis indicated local buckling & distortional buckling failure in all modelled specimens.
- While increasing the yield strength of the section, load carrying capacity also increased.

REFERENCE

[1] ABAQUS(2013). Dassault systems simuliacorp, ABAQUS standard user’s Manual version 6.13. USA.

[2] AISI – S100:2012 North American specification for the design of cold-formed steel structural members specification , Washington , DC,USA, (2007).