VERIFICATION OF NUMERICAL WITH EXPEREMENTAL RESULTS FOR TAPERED STEEL FRAMES

Abbas. H, Salem. E. and Hamouda. A

ABSTRACT

This paper presents a comparative study between experimental results available in literature and a numerical analysis carried out by the authors dealing with the behavior of tapered steel frames. The effects of nonlinear stress-strain relationship, initial imperfections, and end-restraints (boundary conditions) are taken into consideration. ABAQUS is used for a nonlinear finite element model study. An experimental from literature test was used to verify the results obtained from ABAQUS.

Keywords: Tapered steel frames; Nonlinear analysis; Finite elements; Imperfections,

1- INTRODUCTION

Steel building systems are widely used in industrial building construction for economic causes. Tapered steel frames are used to optimize the cost of these buildings. This paper contains a detailed description of the frames tested by Hong (2007). Comparisons of the test results with analytical results are also provided. A full scale testing of tapered frames was used to provide experimental results for verification of the analytical model, and to examine the importance of nonlinear effects which should be added in further theoretical research.

2- FINITE ELEMENT MODEL

The commercially general-purpose finite element program [ABAQUS, 2001] was used in this study to model local buckling behavior of tapered steel frames. The purpose of these finite element simulations is to accurately predict both the strength and ductility of these frames as they are influenced by flange and web local buckling. A full scale tapered frame tested by Hong (2007) was used to calibrate the finite element model, as shown in Fig. (1). The web and flanges member properties are shown in Table (1). The failure mode of this frame was an interaction of local flange instability, web local instability, and lateral instability. To accurately model these failure modes, the nonlinear geometry and nonlinear material capabilities of the [ABAQUS, 2001] program were used. The shell element used in the model is a general-purpose shell element, SR4, which can provide accurate solution for both thin and thick shell problems. In the formulation of these elements, the change in thickness as a function of in-plane deformation is also included. Four elements were used across the flange and eight elements across the web. The frame was restrained out of plane at the knee bracing locations shown in Figs. (1 and 2). Experimental full scales testing of tapered steel frames were performed by [Miller 2003, Jun Li 2002, and Hong 2007].

3-TESTED MODEL

A steel building with a dimension of 115.3m2 (18.3mx6.3m) containing two frames tested by [Hong, 2007] was used to verify the finite element model. The span of the tested frame is 18.29m, the height of the frame columns is 6.096m and the roof slope was 1/24, more details can be found in [Jong-Kook Hong, 2007], as shown in Fig. (1).

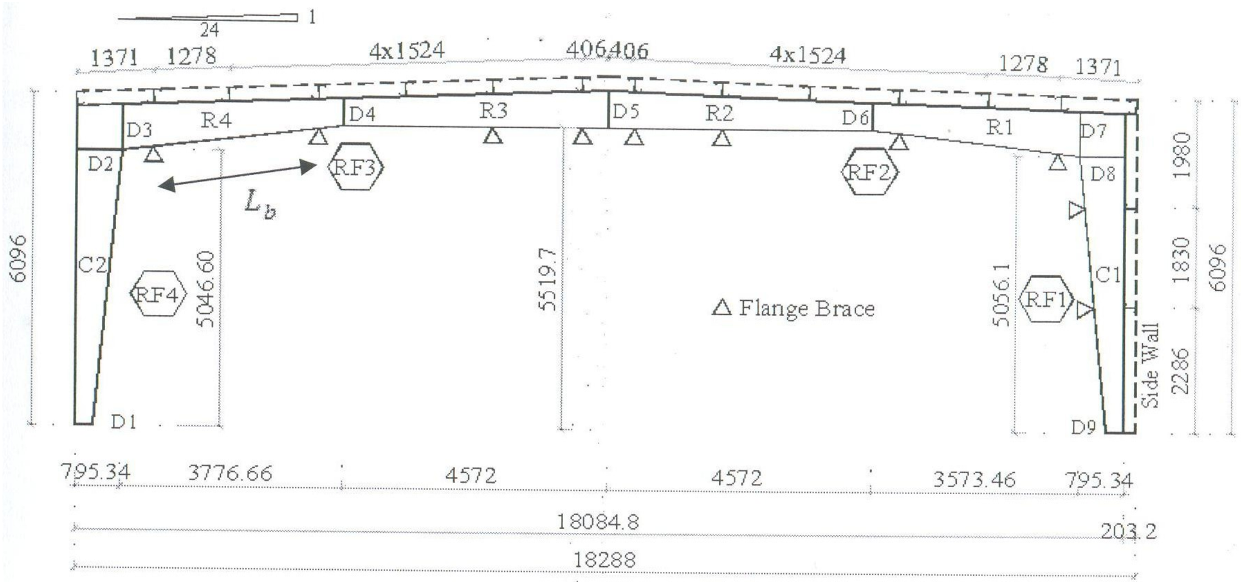

Fig. 1-Tested model dimensions (mm) (Hong 2007)

Table 1- Member Properties

Section	h_w start (mm)	h_w end (mm)	tw (mm)	b_f inside (mm)	t_f inside (mm)	b_f outside (mm)	t_f outside (mm)
D1 – D2	304.80	787.40	5.08	203.20	9.525	203.20	6.35
D2 – D2	787.40	787.40	7.95	203.20	9.525	203.20	7.937
D3 – D4	812.80	558.80	5.715	152.40	6.35	152.40	6.35
D4 – D5	558.80	685.80	4.445	152.40	6.35	152.40	6.35
D5 – D6	685.80	558.80	4.445	152.40	6.35	152.40	6.35
D6 – D7	558.80	812.80	5.715	152.40	6.35	152.40	6.35
D8 – D8	787.40	787.40	7.95	152.40	7.937	152.40	9.525
D8 – D9	787.40	304.80	5.08	152.40	6.35	152.40	7.937

Fig. 2- Frame Loading (Hong 2007).

4- Example of Member Details (RF4)

Properties of all sections of the tested frame are given in table (1). All data for elements are written from left to right (RF4 - RF3 – RF2 – RF1), i.e. start of element and end of element are written from left to right (D1 to D9). For example, member properties for (RF4) are shown in Fig. (3).

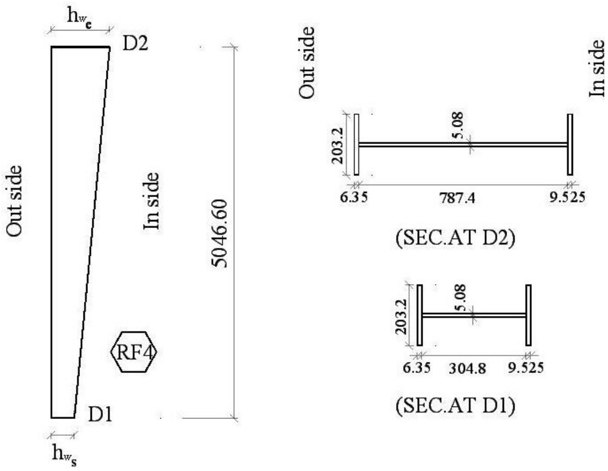

Fig. 3- Detail of Member RF4

5- STEEL MATERIAL PROPERTIES

The properties of the materials used are shown in Tables 2 and 3. The Nominal yield stresses (σ_y) for web and flanges are 395 MPa and 360 MPa, respectively. Engineering stress-strain and true stress strain curves, for web and flanges, are shown in Figs. (4 and 5). The data points shown on the nominal stress-strain curve will be used to determine the plastic data for web and flanges, respectively.

Table 2- Web Plate Stress-Strain Curve Data Points

Point	Nominal Stress (MPa)	Nominal Strain	True Stress (MPa)	True Strain
1	0.00	0.0000	0.000	0.0000
2	395.00	0.00197	395.780	0.0020
3	395.00	0.0250	404.875	0.0250
3	470.00	0.0600	498.200	0.0580
5	510.00	0.1000	561.000	0.0950
6	525.00	0.1500	603.750	0.1400
7	525.00	0.2000	630.000	0.1820

Table 3- Flange Plate Stress-Strain Curve Data Points

Point	Nominal Stress (MPa)	Nominal Strain	True Stress (MPa)	True Strain	Plastic Strain
1	0.00	0.0000	0.00	0.0000	0.0000
2	360.00	0.0018	360.65	0.0018	0.0001
3	360.00	0.0200	367.20	0.0198	0.0181
3	410.00	0.0300	422.30	0.0296	0.0275
5	470.00	0.0600	498.20	0.0583	0.0559
6	500.00	0.1000	550.00	0.0953	0.0927
7	512.00	0.1500	588.80	0.1398	0.1370
8	512.00	0.2000	614.40	0.1823	0.1794

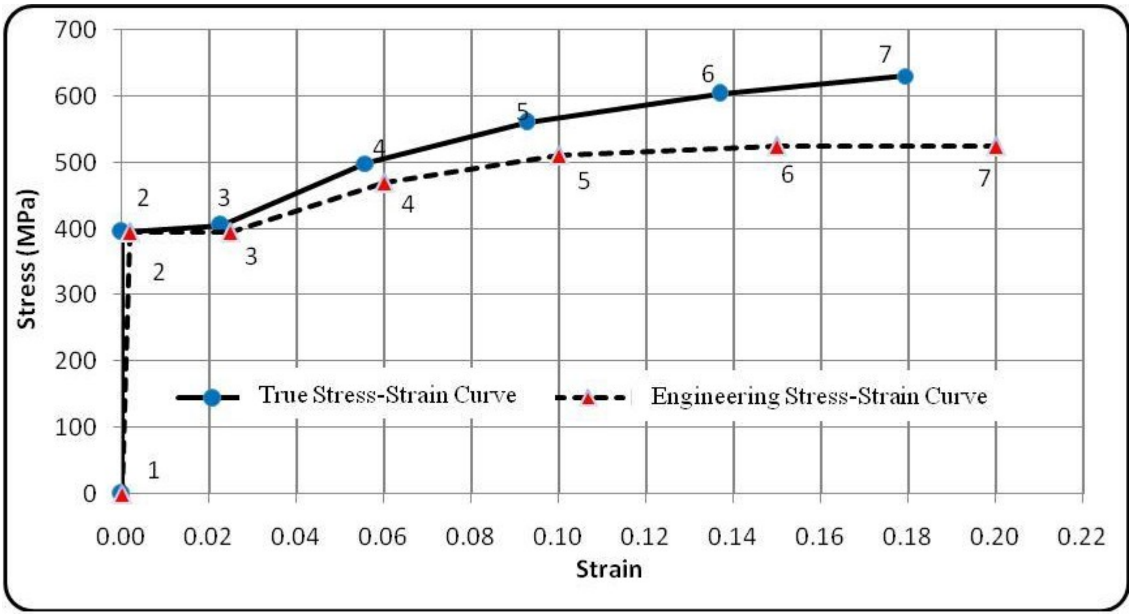

Fig. 4- Engineering and True Stress – Strain Curves for Web Plate

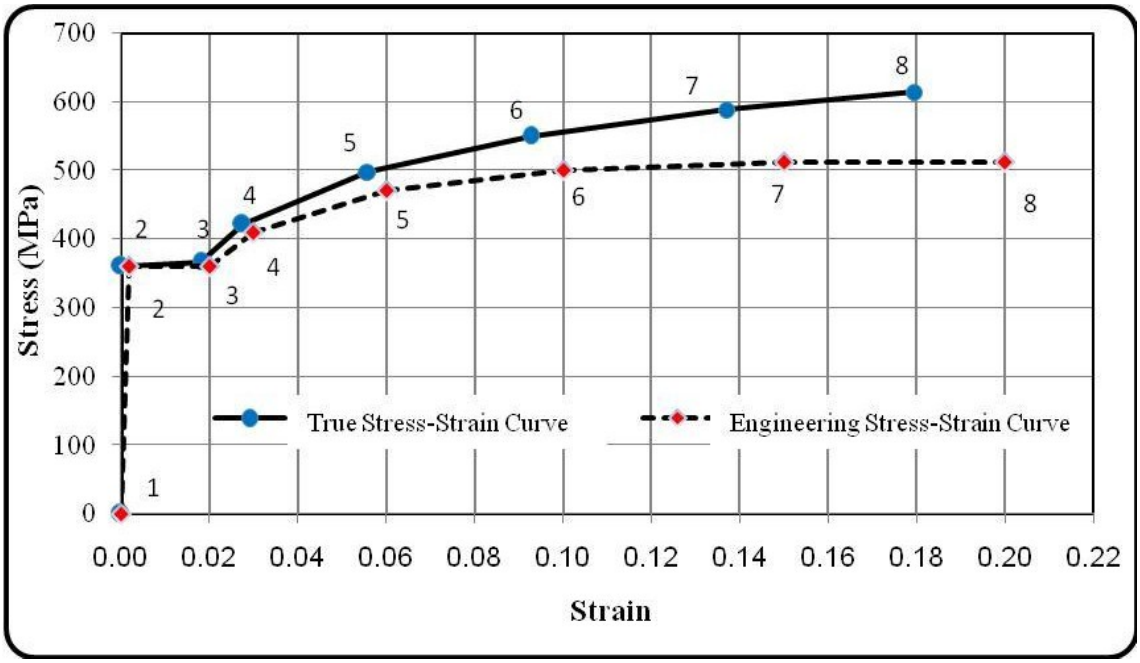

Fig. 5- Engineering and True Stress – Strain Curves for Flange Plate

6- TEST ARRANGEMENT

Gravity load component of 2.52 kN/m' was applied on frame in the first step as uniform loads on flange. In the second step, a horizontal load of 198.75 kN was applied at column top as shown in Fig. (6).

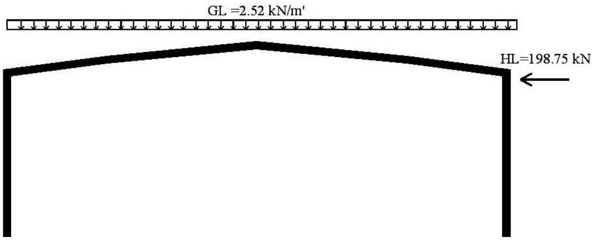

Fig. 6-Frame Loading

In Hong's experiment, upon applying the gravity load, maximum vertical deflections at mid-span of 5.6 mm and 6.1 mm were observed for frames 1 and 2, respectively. The ABAQUS model reaches a total gravity load and deflects at the frame mid-span by10.6 mm. and 10.8 mm. for frames 1 and 2 respectively. The horizontal load reached a total value of 199.25kN before unloading and had a maximum horizontal deformation at the top of the column of 138.9 mm. The ABAQUS model reached a total load of 205.94kN before unloading and a maximum horizontal deformation at the top of column of138.18mm. Figure (7) shows a comparison between the deflections obtained experimentally and analytically. The results of the two models are almost similar as the two lines are approximately parallel throughout the tests. Plastic local buckling deformations were observed adjacent to the knee beam-to-column connection and the ridge beam-to beam connection, as shown in Figure (8).

The relationship between the horizontal displacement and horizontal jack load of the tested frame is given in Fig. (7).

In the analytical model, the buckling deformation is similar to the experimental one, as shown in Fig. (9).

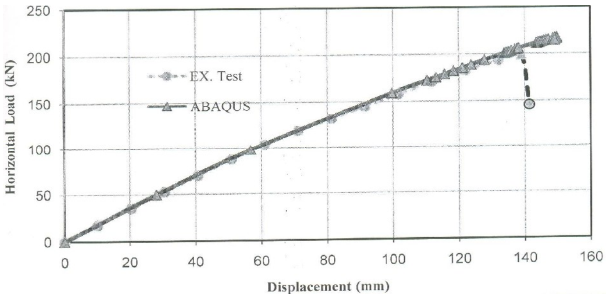

Fig. 7- Column Top Displacements (no imperfections

(a) Frame 1 Failure Mode **(b) Frame 2 Failure Mode**

Fig. 8- Failure Modes (Experimental Test) (Hong 2007)

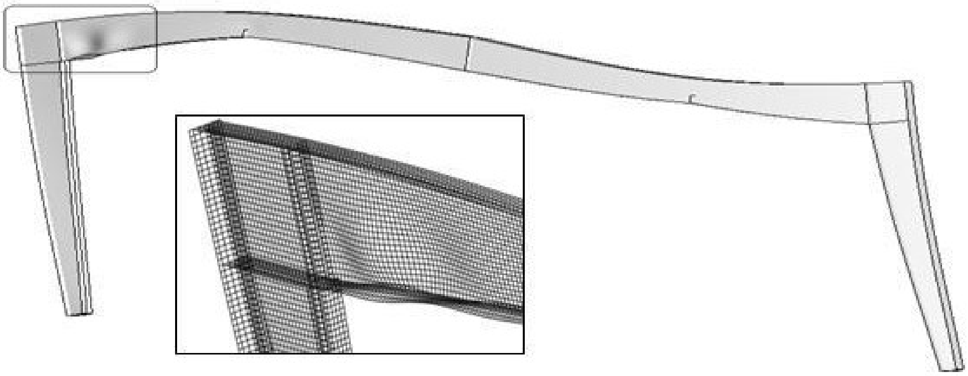

Fig. 9- First Buckling Mode Shape (Analytical Model)

7-INITIAL GEOMETRIC IMPERFECTIONS

Introducing imperfections is necessary in performing nonlinear analysis. To generate the imperfected shape, four steps are required. Firstly, two concentrated loads were applied at the location of maximum compressive stress in the lower flange as shown in Fig. (10). Secondly, the deformed shape from static analysis is generated as shown in Fig. (11). thirdly, the node displacements are added to the original joint coordinates. Finally, the new model joint coordinates are imported to ABAQUS to generate the imperfected shape.

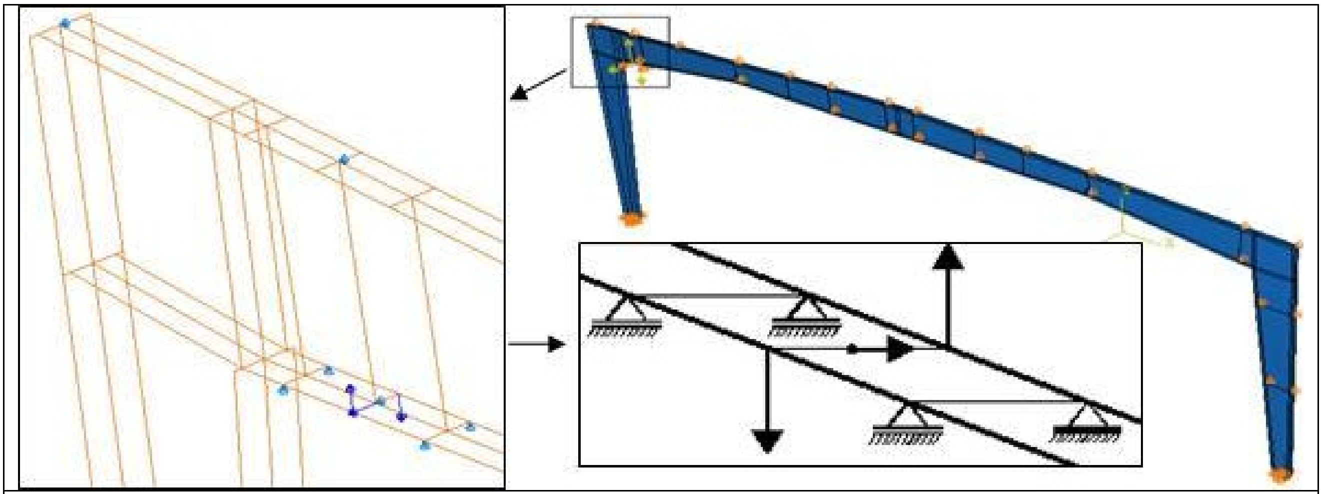

Fig. 10- Imperfection Load in ABAQUS Model

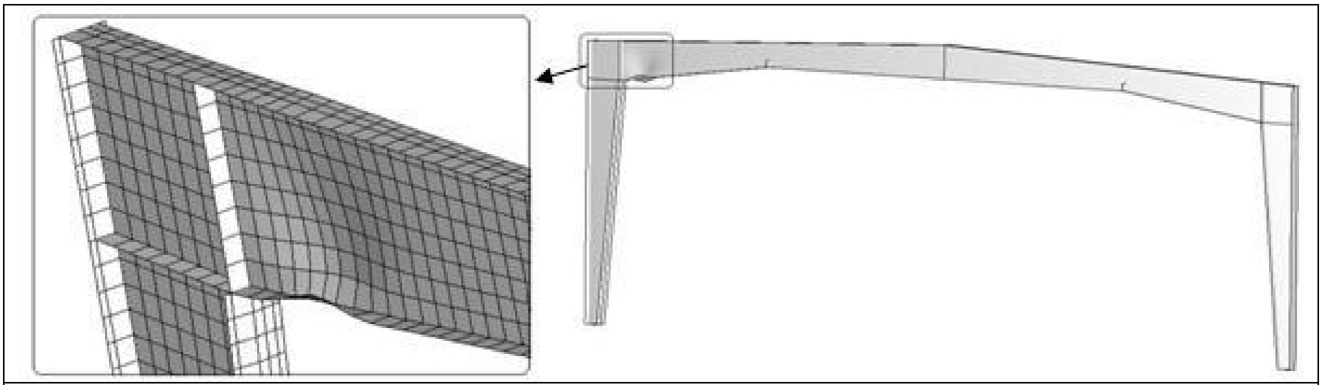

Fig. 11- Imperfection Seed in ABAQUS Model

8-COMPARISON BETWEEN EXPERIMENTAL AND FINITE ELEMENT RESULTS

The finite element simulation (using ABAQUS) shows good agreement with the experimental results of the frame tested by [Jong-Kook Hong 2007]. The column top lateral displacement and the corresponding lateral load are shown in Figs (12, 13, 14, 15 and16), for both experimental and finite element simulation for different imperfection amplitudes related to L_b where L_b is the unbraced length of the member. The von Mises stress contours at peak load are shown in Fig. (16-a,b).

Different imperfection amplitudes were selected between (L_b/500) and (L_b/5000), and their effect on the ultimate loads are shown in Figs (12, 13, 14, 15 and16-a,b). Table (4), summarizes the effect of the imperfection amplitude on the ultimate load.

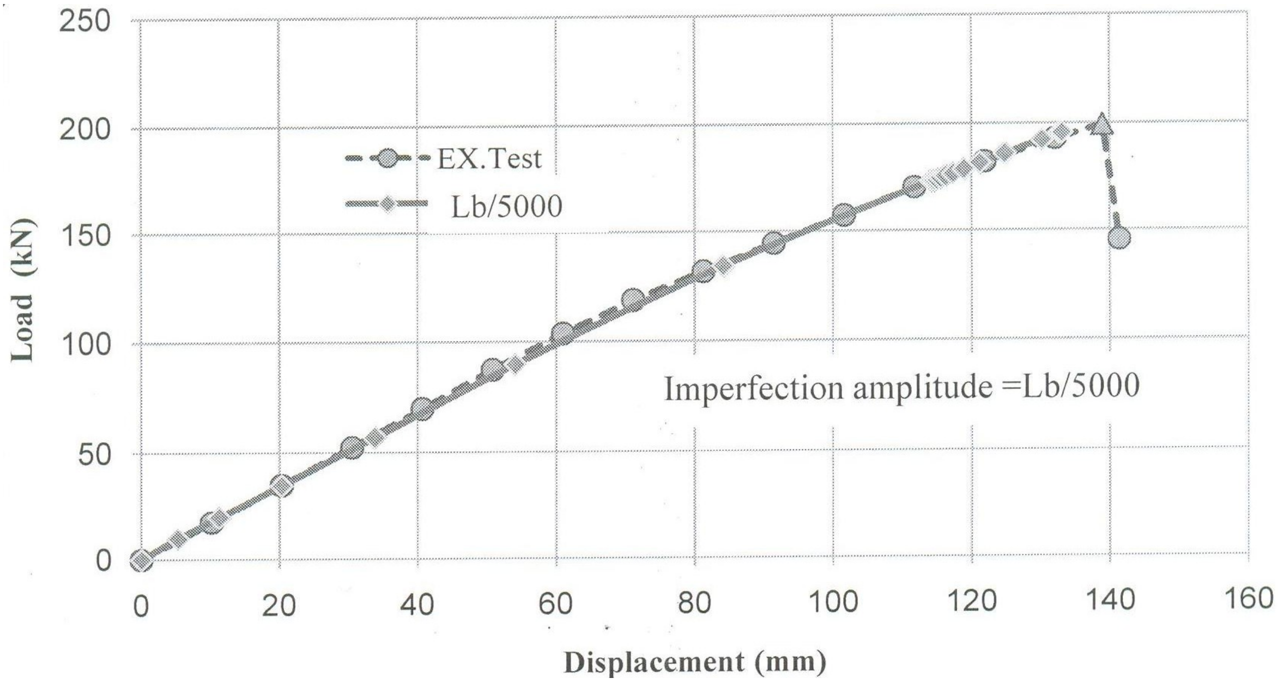

Fig. 12- Column Top Displacements (imperfection amplitude = Lb/5000)

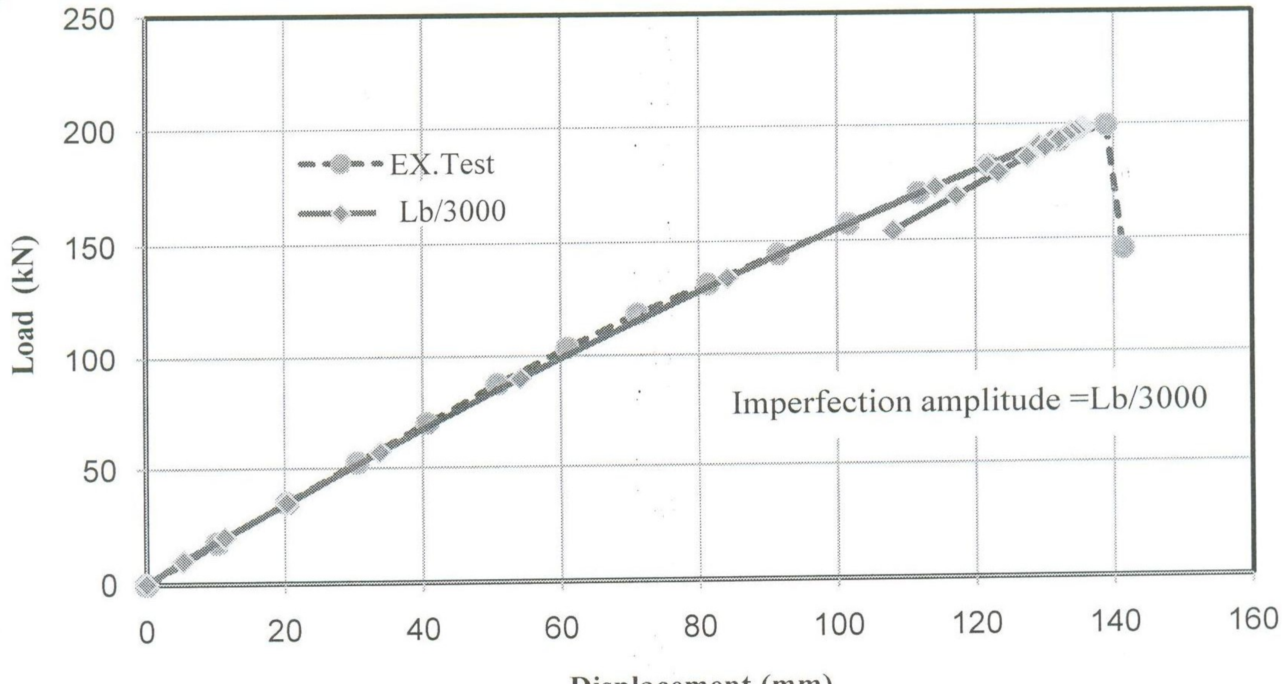

Fig. 13- Column Top Displacements (imperfection amplitude = Lb/3000)

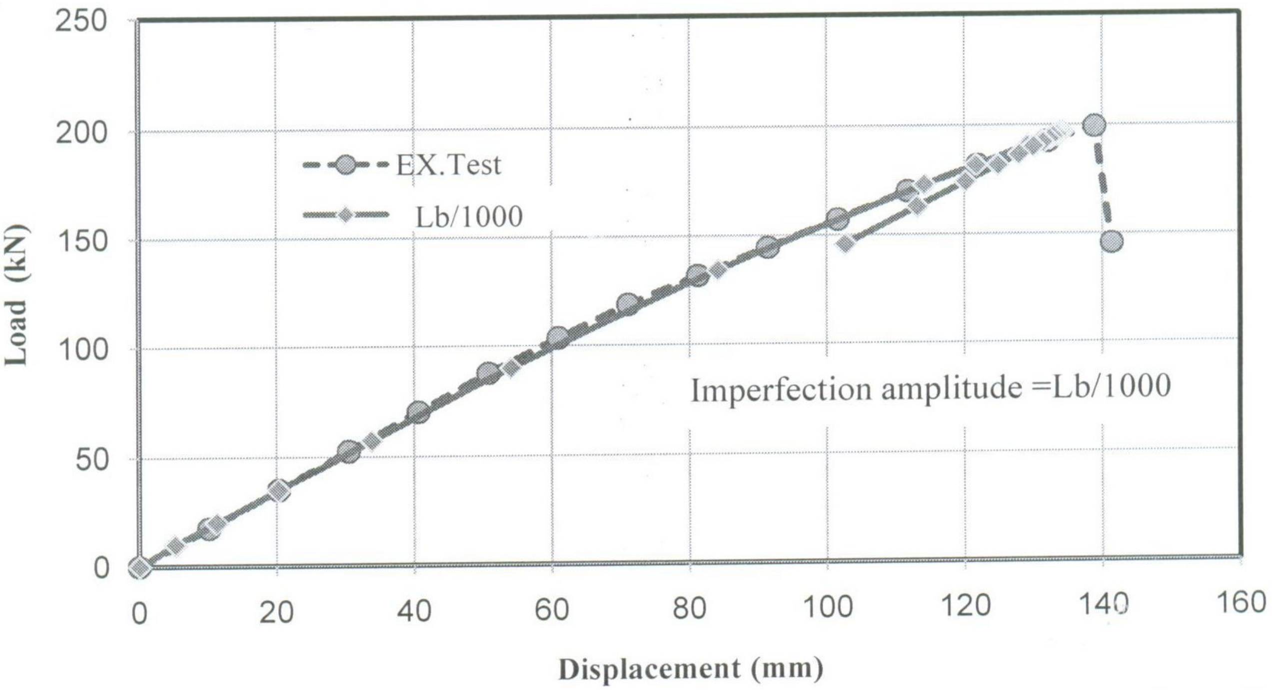

Fig. 14- Column Top Displacements (imperfection amplitude = Lb/1000)

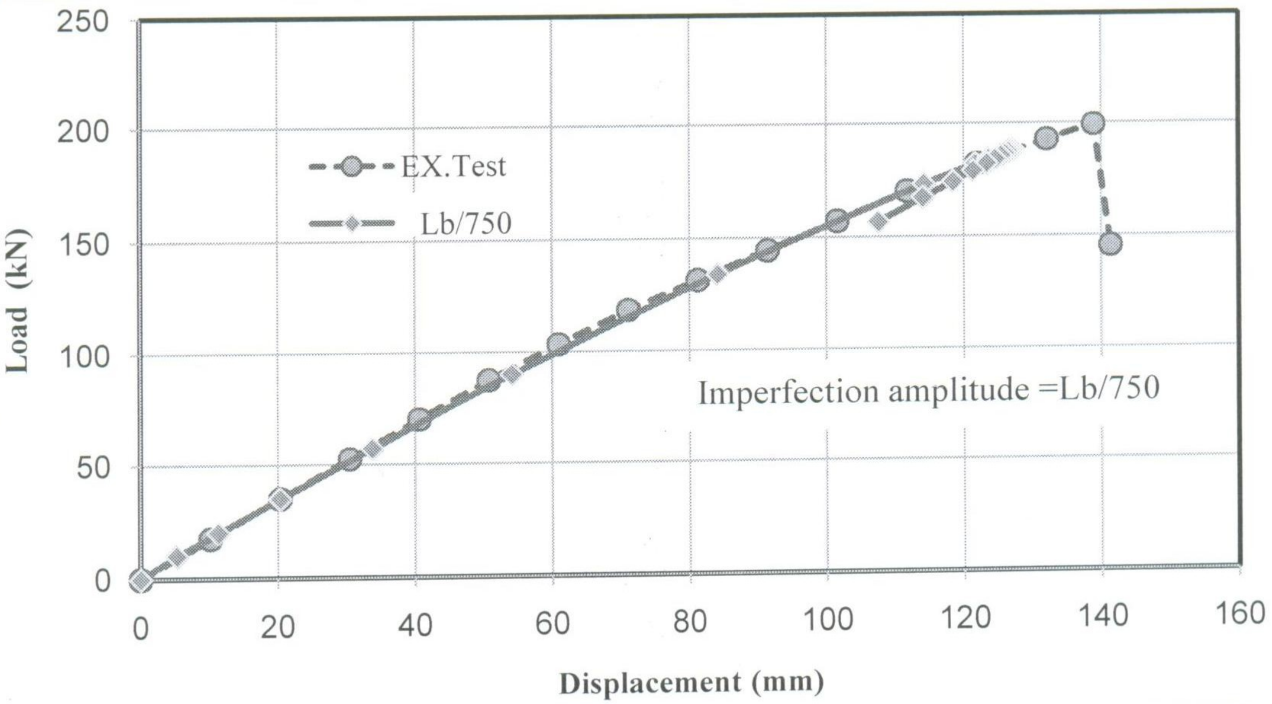

Fig. 15- Column Top Displacements (imperfection amplitude = Lb/750)

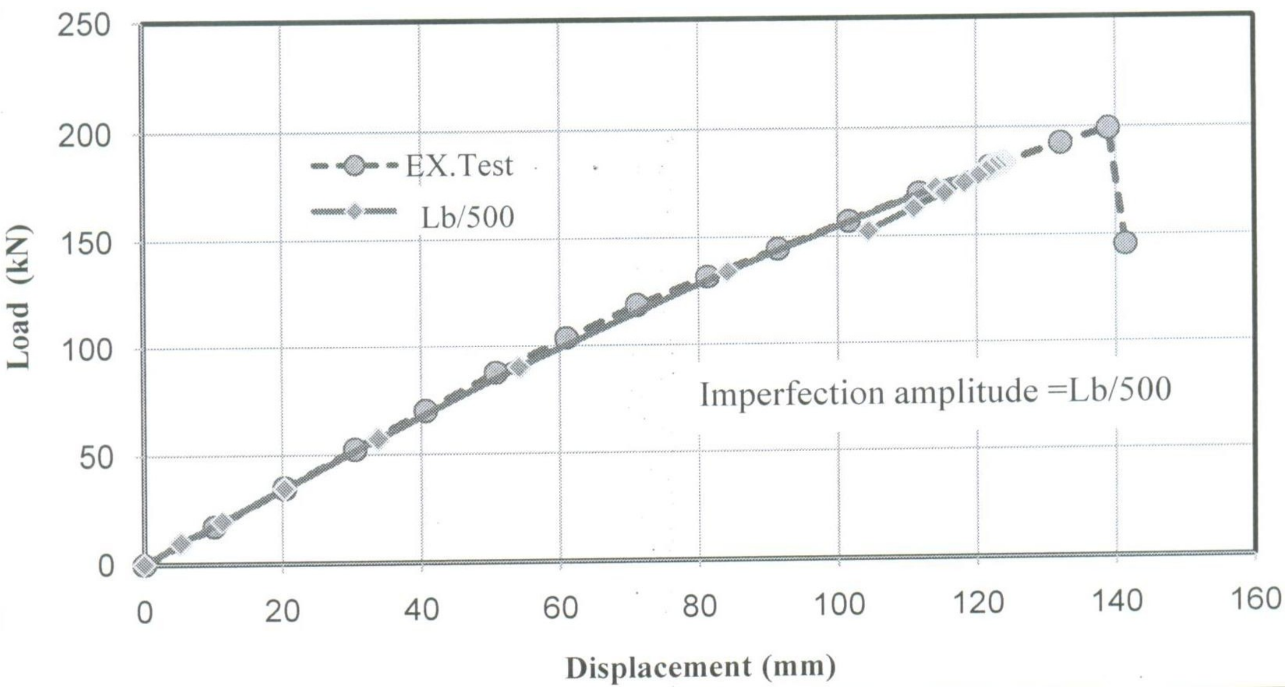

Fig. 16-A- Column Top Displacements (imperfection amplitude = Lb/500)

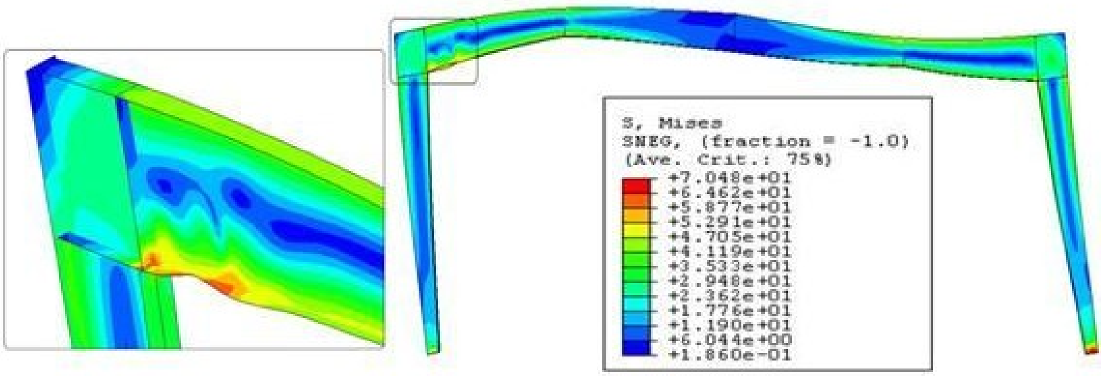

Fig. 16-B- Von Mises Stress Contour at Peak Load

Table 4- Imperfection Amplitude vs. Ultimate Load

	Lb/500	Lb/750	Lb/1000	Lb/3000	Lb/5000
Imperfection Amplitude (mm)	5.60	3.70	2.80	0.94	0.56
Ultimate Load (kN)	184	188	192	197	199

9- CONCLUSIONS

The following conclusions are arrived at, from this present work:

1- Approximate assumption of initial geometric imperfections was necessary to predict the frame behavior. Good results were reached with initial geometric imperfections introduced at the expected failure location.

2- Parametric studies show that the best result is achieved when using initial geometric imperfection equal to $L_b/1000$, where L_b is the unbraced length of the member.

3- Increasing imperfections amplitude results in reducing the ultimate load.

4- Distributed plasticity analysis (ABAQUS) accurately predicts the experimental behavior.

REFERENCES

1- *ABAQUS, (2001) ABAQUS Standard User's Manual, Version 6.3, Volumes 1 to 3, Hibbit, Karlsson & Sorensen, Inc., Pawtucket, Rhode Island, USA.*

2- *ABAQUS, (2001) ABAQUS Theory Manual, Version 6.3, Hibbit, Karlsson & Sorensen, Inc., Pawtucket, Rhode Island, USA.*

3- *Jong-Kook Hong [2007] "Development of A Seismic Design Procedure for Metal Building Systems", University of California, San Diego*

4- *Jun Li and Guo-Qiang Li*[2002] "Large-scale Testing of Steel Portal Frames Comprising Tapered Beams and Columns" Advances in Structural Engineering Vol. 5 No. 4*

5- *Miller, B. S. and Earls, C. J. [2003] "Behavior of Web-Tapered Built-up I Shaped Beams", Report CE/ST 28, University of Pittsburgh, Pittsburgh, PA.*

6- *Abbas et al. [2013] "Nonlinear Analysis of Steel Frames: Comparison Between Some Design Methods," Journal of Al-Azhar University, Engineering Sector, JAUES, Vol. 8, No. 26, January 2013, Cairo, Egypt.*

7- *Hamouda Ayman. [2013] "Nonlinear Analysis of Steel Frames", M. Sc. Thesis, Faculty of Engineering, Al Azhar University, Cairo, Egypt*