Nonlinear Analysis of the Gusset Plate Connection for Braced Frames Subjected to Monotonic and Cyclic Loading

Mohamed B. Abel Wahab1, *, Mohamed A. Galal1, Emad S. Salem1, 2

1Civil Engineering Department, Faculty of Engineering, Al-Azhar University, Cairo, Egypt
2Construction Engineering, Faculty of Engineering, Galala University, Suze, Egypt

*Correspondence: ceng.muhammedbahaa1990@gmail.com

ABSTRACT

In this paper, studies on braced frames are presented to study their behavior under the influence of seismic loads. Steel braced frame systems are considered an efficient and economical technique to resist horizontal and maintain the lateral deformation of buildings under the effect of wind and seismic loads. Among these studies, a study was conducted at the University of California - Berkeley, America, in 2012-2013 by Lai and Mahin. This study conducted laboratory tests for three full-scale one-bay two-story braced frames with different cross sections for the braces such as square-HSS, circular-HSS and wide-flange I beam. A three-dimensional model is conducted to validate the laboratory test, using ABAQUS program. In this model, non-linear material properties are used for each part of the model. Two types of loading are developed, nonlinear static and nonlinear dynamic. A comparison is made between the results of the experimental test and the numerical model, which showed that good agreement between these results.

KEYWORDS: Experimental test, Finite element modeling, Monotonic loading, Cyclic loading, braced frames, gusset plate connections.
NONLINEAR ANALYSIS OF THE GUSSET PLATE CONNECTION FOR BRACED FRAMES SUBJECTED TO MONOTONIC AND CYCLIC LOADING

الملخص العربي:-
في هذا البحث، تم تقديم الدراسات التي أجريت على الأطارات الفراغية ذات الشكلاء لدراسة سلوكها تحت تأثير أحمال الزلازل. ومن هذه الدراسات، دراسة أجريت في جامعة كاليفورنيا - بيركلي بأميريكا عام 2012 - 2013 بواسطة كلا من لاي وماهين. وقد قام في هذه الدراسة باختبارات معتمدة لثلاث نماذج كاملة بقطاعات مختلفة للأشكال التي تدخل هذه الإطارات. وقد تم عمل شرائح ثلاثية الأبعاد لأحد هذه النماذج التي تم استخدامها في الدراسات المعملية بواسطة برنامج الأباكس. وقد تم التأثير على النموذج بنوعين من الأحمال، أحدهما خطي والأخر دوري. وقد تم عمل مقارنة بين نتائج الأختبار العملية والنموذج العدجي التي أظهرت اتفاقا جيدا بين هذه النتائج.

الكلمات المفتاحية : الأختبار المعملي، نموذج العناصر المحدودة، الأحمال الأحادي، الأحمال الدورية، الأطارات ذات الشكلاء، ووصلات أواخر النقوية.

1. INTRODUCTION

Steel braced frame systems are considered an efficient and economical technique to resist horizontal and to maintain the lateral deformation of buildings under the effect of wind and seismic loads. There are many shapes of braced frames such as single diagonal, chevron, diamond, single-X, stacked-X and split-X as shown in Fig. 1. Braced frames divided into three main types according to configuration [1]: Concentrically braced frames (CBF), Non-concentrically braced frames (NCBF) and Eccentrically braced frames (EBF) as shown in Fig. 2 and Fig. 3. The buckling restrained braced frame (BRBF) is a special type of CBF because the braces do not buckle when loaded in compression. Special concentrically braced frame (SCBF) is a type of (CBF) with additional detailing requirements to provide high energy dissipation capacity and to resist ductility under inelastic deformations. One of these requirements is the connections that should be designed on the force demands that obtained from the maximum expected forces of the brace in tension and compression [1]. There are many researchers conducted their research on the components of the braced frames to improve the behavior of these braced frames under highly seismic zones.

One of the components of the braced frames is the bracing member. The bracing member will buckle under compression loads and yield under tension loads, which make the behavior of these member’s complex. There are many research study the performance of multi-story braced frame either numerical or experimental have been conducted on the multi-story braced frames.

The second components of the braced frames is the connections. One of these connections are the connections between the brace member to the beam and the column which called gusset plate connections. Many studied the connections performance in terms of the distribution of forces as well as the buckling of the gusset plate under compression loads and yielding under tension loads.

which formed by AISC and he developed the Uniform force method. Ricker DT 1992 [8] presented the Parallel force method. Thornton WA. 1992 [9] proposed the KISS method (Keep It Simple and Stupid). Astaneh-Asl A. 1998 [10] presented the Concentric force method. Muir LS 2008 [11] proposed the Generalized Uniform Force Method (GUFM) in order to eliminate the constraint that exists in the (UFM). A series of the experimental and analytical investigations has been conducted in University of Washington from 2005 - 2011 to improve and enhance the seismic performance of SCBFs. The research program included single-story, two-story, and three-story tests. Also, more than 30 single-story, single-bay SCBFs were tested to investigate the connection details of the corner gusset plates and present a new balanced design procedure with an elliptical clearance model for the corner gusset plates [12][17]. Ebrahimi et al. 2019 [18] proposed a design procedure for the gusset plate dimensions and force distribution at the interface between the gusset plate to beam and column.

Fig. 1 : Different shapes of braced frames system [1].

Fig. 2 : Typical concentric gusset connection and nonconcentric brace work point [1].
2. DESCRIPTION OF THE EXPERIMENTAL BRACED FRAME

The test specimen consists of one bay two stories Special concentrically braced frame with the height of stories 9 ft and the span of 20 ft as shown in Fig. 4. Fig. 5 shows the types of members for the test specimen the specimen was tested quasi-statically under cyclic loading. All braces were slotted and welded to the gusset plates as per AISC requirements. Based on these requirements, reinforcing plates were welded to the braces at the net reduced section of the braces at end of the gusset plate.

Fig. 3 : Eccentrically braced frame [1].

Fig. 4 : Overview and dimension of test specimen TCBF-B-1 [19].
Complete joint penetration weld details were specified at the lower beam-to-gusset plate connections. The beam top and bottom flanges were 45° grooved, and welded to gusset plate finger stiffeners. Backing bars were used on the top flange and bottom flange welds and not removed after welding. The beam web was also 45° grooved and welded on one side as shown in Fig. 6.
3. FE MODELING

A three-dimensional finite element model for braced frame was developed to simulate the observed experimental test conducted by (Jiun-Wei Lai; Stephen A. Mahin, 2013) [19] using ABAQUS/CAE (version 6.14-4) [20]. Detailed description of the finite elements used, material properties, element type, boundary condition and a comparison between the finite element and experimental results are presented in the following sections.

3.1. MODEL GEOMETRY

Braced Frame Model contains two steel columns, six beams with different lengths, four braces and other parts to tie the columns to the beams. The span between columns 6140 mm (c/c), first story height 2760 mm (c/c) and second story height 2780 mm (c/c) as shown in Fig. 8 and Fig. 9. Basic dimensions and engineering properties of sections used for beams, columns, braces and other parts are listed in Table 1 to Table 3 [1].

Fig. 8: Model geometry [19].
Fig. 9: Details 1, 2 and 3 for the model geometry [19].
NONLINEAR ANALYSIS OF THE GUSSET PLATE CONNECTION FOR BRACED FRAMES SUBJECTED TO MONOTONIC AND CYCLIC LOADING

Table 1: Dimensions of columns and beams.

<table>
<thead>
<tr>
<th>Part No.</th>
<th>Part name</th>
<th>b_f (mm)</th>
<th>t_f (mm)</th>
<th>d_w (mm)</th>
<th>t_w (mm)</th>
<th>Length (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Column W(12X96)</td>
<td>330</td>
<td>22</td>
<td>330</td>
<td>14</td>
<td>6400</td>
</tr>
<tr>
<td>2</td>
<td>Upper Beam W(24X117)</td>
<td>330</td>
<td>22</td>
<td>616</td>
<td>14</td>
<td>5766</td>
</tr>
<tr>
<td>3</td>
<td>Lower Beam W(24X68)</td>
<td>228</td>
<td>14</td>
<td>603.25</td>
<td>11</td>
<td>4406.9</td>
</tr>
<tr>
<td>4</td>
<td>Lower Small Beam W(24X68)</td>
<td>228</td>
<td>14</td>
<td>603.25</td>
<td>11</td>
<td>679.55</td>
</tr>
<tr>
<td>5</td>
<td>Upper Stub Beam W(24X117)</td>
<td>330</td>
<td>22</td>
<td>616</td>
<td>14</td>
<td>500</td>
</tr>
<tr>
<td>6</td>
<td>Lower Stub Beam W(24X68)</td>
<td>228</td>
<td>14</td>
<td>603.25</td>
<td>11</td>
<td>500</td>
</tr>
</tbody>
</table>

Table 2: Dimensions of braces.

<table>
<thead>
<tr>
<th>Part No.</th>
<th>Part name</th>
<th>Outer width (mm)</th>
<th>Outer depth (mm)</th>
<th>Thickness of box (mm)</th>
<th>Length (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>Upper brace HSS(5X5X5/16)</td>
<td>120</td>
<td>120</td>
<td>8</td>
<td>2413</td>
</tr>
<tr>
<td>8</td>
<td>Lower brace HSS(6X6X3/8)</td>
<td>150</td>
<td>150</td>
<td>9</td>
<td>2751</td>
</tr>
</tbody>
</table>

Table 3: Dimensions of plates and stiffeners.

<table>
<thead>
<tr>
<th>Part No.</th>
<th>Part name</th>
<th>Width of plate b_p (mm)</th>
<th>Depth of plate d_p (mm)</th>
<th>t_p (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>Upper and lower stub plate</td>
<td>780</td>
<td>880</td>
<td>50</td>
</tr>
<tr>
<td>10</td>
<td>Base plate at column</td>
<td>800</td>
<td>720</td>
<td>50</td>
</tr>
<tr>
<td>11</td>
<td>Base plate at middle</td>
<td>1520</td>
<td>650</td>
<td>50</td>
</tr>
<tr>
<td>12</td>
<td>(8) Stiffener with upper bracing -(c103)</td>
<td>50</td>
<td>355</td>
<td>16</td>
</tr>
<tr>
<td>13</td>
<td>(8) Stiffener with lower bracing -(d103)</td>
<td>75</td>
<td>405</td>
<td>16</td>
</tr>
<tr>
<td>14</td>
<td>(16) Stiffener with column-(k100)</td>
<td>165</td>
<td>330</td>
<td>22</td>
</tr>
<tr>
<td>15</td>
<td>(2) Stiffener with upper beam-(c102)</td>
<td>165</td>
<td>616</td>
<td>12.5</td>
</tr>
<tr>
<td>16</td>
<td>(2) Stiffener with upper gusset plate-(f102)</td>
<td>155.5/107</td>
<td>436.5</td>
<td>12.5</td>
</tr>
<tr>
<td>17</td>
<td>Stiffener with upper gusset plate -(g102)</td>
<td>230</td>
<td>670</td>
<td>12.5</td>
</tr>
<tr>
<td>18</td>
<td>(2) Stiffener with lower beam-(k102)</td>
<td>165</td>
<td>603.25</td>
<td>12.5</td>
</tr>
<tr>
<td>19</td>
<td>(4) Stiffener with lower beam-(g100)</td>
<td>203</td>
<td>518</td>
<td>12.5</td>
</tr>
<tr>
<td>20</td>
<td>Stiffener with base plate at middle-(h103)</td>
<td>105.5/150.5</td>
<td>500</td>
<td>12.5</td>
</tr>
<tr>
<td>21</td>
<td>Stiffener with base plate at middle-(j103)</td>
<td>115</td>
<td>230</td>
<td>12.5</td>
</tr>
<tr>
<td>22</td>
<td>All gusset plates</td>
<td>See Details</td>
<td></td>
<td>19</td>
</tr>
</tbody>
</table>
3.2. MATERIAL PROPERTIES

Braced Frame Model contains on four types of steel materials which are ASTM A992, ASTM A572 Grade 50, ASTM A500 Grade B and ASTM A36. All wide flange beams and wide flange columns were made of ASTM A992 steel section. All braces using hollow structural sections (HSS) were made of ASTM A500 Grade B steel tubes for square sections. The gusset plates, base plates, stub beam end plates, shear tabs, finger plates, continuity plates and brace reinforcing cover plates were made of ASTM A572 Grade 50 steel plates. The beam web stiffener plates, lifting lugs, shim plates, and miscellaneous parts were made of ASTM A36 steel plates [19]. The elastic properties of this model are; Young’s modulus, \( E \), is 200,000 MPa and Poisson’s ratio, \( \nu \), is 0.3. When defining materials in ABAQUS [20], the elastic and inelastic material behaviour must be defined. Modulus of elasticity for steel is defined to predict the behaviour of material in the elastic range, while the true stress (\( \sigma_{\text{true}} \)) and logarithmic plastic strain (\( \varepsilon_{\text{pl}} \)) are required to define the material after the elastic range.

The following equations are used to convert engineering stress – strain curve to true stress and plastic strain curve in ABAQUS 6.14-4:

\[
\sigma_{\text{true}} = \sigma_{\text{nom}} (1 + \varepsilon_{\text{nom}}) \\
\varepsilon_{\text{true}} = \ln (1 + \varepsilon_{\text{nom}}) \\
\varepsilon_{\text{pl}} = \varepsilon_{\text{true}} - \varepsilon_{\text{el}} = \varepsilon_{\text{true}} - \sigma_{\text{true}} / E
\]

Where:
- \( \sigma_{\text{true}} = \) true stress, \( \sigma_{\text{nom}} = \) engineering (nominal) stress, \( \varepsilon_{\text{nom}} = \) engineering (nominal) strain, \( \varepsilon_{\text{pl}} = \) plastic strain, \( \varepsilon_{\text{true}} = \) true strain, \( \varepsilon_{\text{el}} = \) true elastic strain, \( E = \) Young’s modulus.

The previous equations were used for all materials defined in the finite element model as shown in Fig. 10.
3.3. ELEMENT TYPE AND MESH SIZE

A four-node shell element, S4R with reduced integration in ABAQUS 6.14-4 [20] was used for all instances to create the finite element model. The S4R element has six degrees of freedom per node, quadrilateral element that is suitable for a wide range of applications and its better convergence rate for the section points through the shell thickness. Before assigning mesh instances, each instance is seeded using seed part technique. For models with complex details, the meshing quality is important. Duplicate mesh elements are needed to get accurate results from a finite element simulation, so it is sometimes necessary to partition the parts to define a better mesh quality. The mesh sizes of the parts are often related to each other. A finer mesh is used in the regions of high stresses or deformation gradients, while a coarser mesh is used in regions of low stresses or deformation gradients. Mesh is refined in gusset plates and bracing members where the stress concentration and deformation are expected as shown in Fig. 11 and Fig. 12.
Fig. 11: Typical mesh size for ABAQUS model.

Fig. 12: Finer mesh for gusset plate connections.
3.4. **BOUNDARY CONDITIONS**

Boundary conditions in ABAQUS were defined first in the initial step and posted to the next step. The FE model was supported at fifth regions according to the experimental test as shown in Fig. 13. The first and second regions are supported in ABAQUS in six directions (fixed support) and applied at the end of columns. The third region is supported in six directions (fixed support) and applied at the end of lower bracing. The sixth and seventh regions are restraint in z-direction to prevent lateral buckling for the upper and lower beams as shown in Fig. 14.

![Boundary conditions for the experimental test](image1)

**Fig. 13**: Boundary conditions for the experimental test [19].

![Boundary Conditions in ABAQUS Model](image2)

**Fig. 14**: Boundary Conditions in ABAQUS Model.
3.5. Loading Application and Analysis Method

Loads can be defined by using the displacement-controlled method or the loading-controlled method. When using any two methods, the load is incrementally applied to the node or surface. While there is another type of load which is called the cyclic load. In the cyclic load, the load is applied by a cyclic loading protocol.

Experimental test on the specimen TCBF-B-1 was tested quasi-statically, with a prescribed history of the upper beam displacement imposed. The displacement of the upper beam was monitored and controlled during the entire test process and the lower beam actuator was force controlled. The sign convention for imposed displacements and forces are that positive displacements and forces correspond to the actuator pushing the specimen to the east side of laboratory while negative values correspond to the actuator pulling the specimen to the west side of laboratory [19].

In the FE model, the loads were applied by two methods. The first method was the monotonic loading and the second method was the cyclic loading method. In the first method, the load applied on the lower beam was one-half of the upper beam. The upper load was $2.6688 \times 10^6$ N (2668.8 kN) while the lower load was $1.3344 \times 10^6$ N (1334.4 kN) as shown in Fig. 15.

![Fig. 15: Applied loads as a force load in ABAQUS model.](image)

The time step is used to define the analysis steps and take place in two stages. ABAQUS define the initial step by default to apply boundary conditions, but the second step must then define to apply the loadings. In this research one “Static, Riks” procedure was used, in addition to the initial step. When the loads can cause large deformations, the geometric nonlinearity is important, so the ‘Nlgeom’ option should be turned ‘On’ to consider the nonlinear geometry effect. The maximum number of increments, initial, minimum and maximum increment size in this research were: 100, $1 \times 10^{-3}$, $1 \times 10^{-15}$, and 0.5, respectively.

In the second method, the loading protocol has been used to apply the cyclic loading. To apply the loading protocol, the amplitude option in ABAQUS was used to
define the cycles of the displacement and time as shown in Fig. 16 [19]. The type of analysis method was considered to be of general static. To apply the load amplitude, two boundary conditions were created. These boundary conditions were of ‘Displacement/Rotation’ type and restrain it in (x-direction) in the direction of the load. One of these boundary condition was applied at the upper beam while the second boundary condition was applied at the lower beam. In the upper boundary condition, the load amplitude was selected and multiplied by 1 in the (x-direction). In the lower boundary condition, the load amplitude multiplied by 0.5 in the (x-direction) as the lower load was one-half of the upper beam.

![Cyclic loading protocol](image)

**Fig. 16 : Applied cyclic loading protocol in ABAQUS model [19].**

### 3.6. GEOMETRIC IMPERFECTION

Steel Structure members often face some crookedness or other geometric imperfections as a result of the manufacturing, transporting, handling and erection processes. There are two types of geometric imperfections: local and global imperfections. Local imperfections can be found in any region of steel member which cause this region to be yielding stressed, while the global imperfections are along the member length in any direction. The effect of geometric imperfection is taken by applying eigenvalue buckling analysis and the worst case of local and global buckling modes can be determined. To define geometric imperfections in ABAQUS model there are three ways: as a linear superposition of buckling Eigen modes, from the displacements of a static analysis, or by specifying the node number and imperfection. In this research a linear superposition of buckling Eigen modes way was used. The lowest 50 (positive) buckling loads (eigenvalues) and the corresponding buckling
shapes (Eigen modes) were determined. Fig. 17 shows the first 4 modes which were taken in the FE model.

Fig. 17: Buckling modes from buckling analysis by ABAQUS, where (A) is the first mode, (B) is the second mode, (C) is the third mode and (D) is the fourth mode.
3.7. **VALIDATION OF FE MODEL**

The FE Model was validated by comparing the analytical and experimental results. The FE model was loaded by two different types of loads, one of the loads was monotonic loading and the second was the cyclic loading.

3.7.1. **Comparison between roof displacement and base shear under the monotonic loading.**

Fig. 18 shows the base shear vs. roof displacement for the experimental work and FE results. The FE model in this comparison was under the monotonic loading. As it can be seen from the figure, the FE model was yield at the same load of the experimental test which equal to 3100 kN which corresponds to a value of 25 mm of displacement approximately. After yielding at the same displacement of the value of 50 mm, the base shear of the FE model was less than the base shear of the experimental test by 2.5% approximately.

3.7.2. **Comparison between roof displacement and base shear under the cyclic loading.**

Fig. 19 shows the base shear vs. roof displacement for the experimental work and FE results under the cyclic loading. As it can be seen from the figure, the results of the FE model and experimental work in a good agreement.
3.7.3. Comparison between the cyclic loading and the monotonic loading.

Fig. 20 shows the comparison between the cyclic and monotonic loading for the FE model and experimental work. As it can be seen from the figure, the monotonic and cyclic loading in a good agreement for the elastic range. After the elastic zone, there was a difference for the base shear at the same values for the displacement which equal to 50 mm where the base shear in the monotonic loading was higher than the base shear in the cyclic loading by 21% approximately. This means that the braced frame in case of FE model, if it is affected by a cyclic loading, collapses faster than if it is affected by a monotonic loading.
SUMMARY

Steel braced frame systems are considered an efficient and economical lateral force resisting systems to control the lateral deformation in buildings under the wind and seismic loads. A three-dimensional finite element model for braced frame was developed to simulate the observed experimental test conducted by using ABAQUS/CAE (version 6.14-4). In the FE model, the loads were applied by two methods. The first method was the monotonic loading and the second method was the cyclic loading method. The FE Model was validated by comparing the analytical and experimental results. As it can be seen from the results, the FE model and experimental work in a good agreement.

REFERENCES


