An Analytical Model of Short Steel Box Columns with Concrete In-Fill (Part 1)

Reza Masoudnia, Soroush Amiri, W.H. Wan Badaruzzaman

Department of Civil and Structural, Engineering, Faculty of Engineering and Built Environment, Universiti Kebangsaan Malaysia, Selangor, Malaysia.

Abstract: A nonlinear three-dimensional finite element analysis of short composite box columns is performed by LUSAS finite element software in this paper. The effect of various geometric parameters such as applied axial load and bending moment on different rectangular cross section of steel tube filled with concrete is considered in this analysis. The short columns are pin ended and subjected to loading. The accuracy of the proposed analytical method is established by comparing the results with corresponding experimental values from Anwar Hossain paper in 2003 and the numerical results, including the load-deflection and interaction diagrams, it was conclude, thickness of steel tube has significant influence especially on the strength of columns and finite element software can be sufficiently accurate in predicting the behavior of composite columns.

Key words: Finite element analysis, Short composite box column, Ultimate load capacity, Load-deflection diagram, Interaction diagram.

INTRODUCTION

Concrete filled steel tubes are economic composite columns, and they have been widely used in many countries. The occurrence of local buckling of steel tube is delayed by the restrain afforded the concrete, and the strength of ductility of concrete is enhanced by the confining effect provided by steel tube. The steel tubes serve as permanent formwork for multistory construction base or cast downwards from the top of the columns. Therefore concrete filled steel tubes are becoming popular for high rise contraction (Liew, 2004; Liang, et al., 2006; Schneider 1998; Young-Hwan Choi, 2004; Anwar Hossain, 2003; Soroush Amiri and Reza Masoudnia, 2011; Reza Masoudnia and Soroush Amiri, 2010; Reza Masoudnia and Soroush Amiri, 2011).

Short CFT columns reach their capacity when the steel yields and concrete crushes. Therefore, the maximum compressive strength of short CFT members (called squash load) can be derived by adding the capacity of each material as shown in Equation 1.

\[ P_b = A_f F_y - k f'_c A_c \]  

(1)

The coefficient k in Equation (1) is used for concrete to modify the measured laboratory values for the compressive strength of concrete cylinders. A value of 0.85 or 1.0 has been used for CFT members. A value of 0.85 has been directly adopted for CFT members just to follow the standards for ordinary reinforced concrete. This results in the lower bound strength. On the other hand, many researchers have also suggested a value of 1.0 for CFT members in order to account for the confinement effect. (Lu and Kennedy, 1994; Knowles and Park, 1970; Sabnis 1979; Furlong, 1967; Tomiai and Sakino, 1979; Schneider, 1998; Young-Hwan Choi, 2004; Anwar Hossain, 2003; Liang, Brian Uy, J.Y. Richard Liew, 2004; Liang, Brian Uy, J.Y. Richard Liew, 2007; Reza Masoudnia and Soroush Amiri, 2010; Reza Masoudnia and Soroush Amiri, 2011).

Introduction and Background of Finite Element Analysis:

The finite element software LUSAS Version 14 is used in this study. The non-liner analysis is carried out to predict the ultimate load-deflection behavior and ultimate load carrying capacity of short composite columns (LUSAS help version 14; Soroush Amiri and Reza Masoudnia, 2011).

Finite Element Analysis was probably the most popular numerical method used today. Finite Element Analysis was first developed by (R. Courant, 1943). He utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems. (LUSAS help version 14; Soroush Amiri and Reza Masoudnia, 2010).

By the early 70’s, FEA was limited to the field of aeronautics, automotive, defense and nuclear industries due to the cost of obtaining a computer at that time. In recent years, the increasing availability of high speed computers has caused civil engineers to embrace finite element analysis as a feasible method to solve complex engineering problems. It is common for personal computers at home today, there are more powerful computer than supercomputer 10 years ago. Therefore, the increasing popularity of Finite Element Analysis can be
attributed to the advancement of computer technology. (LUSAS help version 14; Soroush Amiriand Reza Masoudnia 2010; Reza Masoudnia and Soroush Amiri, 2010).

Finite Element Analysis involves sub-dividing the component analyzed into a mesh of finite sized element of simple shapes. Within each element, the variation of displacement is assumed to be determined by using simple polynomial shape functions and nodal displacements. In the analysis, the equations for the stresses and strains are developed in terms of the unknown nodal displacements. To solve this problem, the equations of equilibrium are assembled in a matrix form which can easily be programmed and solved with a computer. After applying the appropriate boundary conditions, the nodal displacements are found by solving the matrix stiffness equation. (LUSAS help version 14; Soroush Amiriand Reza Masoudnia, 2010).

Modeling of the Column:
Cross-section of short composite box columns tested in the past by other researcher Anwar Hossain in 2003, are shown in the figure 1. The short columns were modeled with real length and width of cross-section. A typical finite element mesh adopt for the short composite columns is shown in Figure 2. Incremental displacement load with adopted for the 1 mm was applied in the negative Z direction and the load acts vertically to the column, simulating the load applied to the columns in experiments. (Anwar Hossain 2003; Soroush Amiri and Reza Masoudnia 2010; LUSAS help version 14).

![Fig. 1: Cross-sections of short composite columns which tested by Anwar Hossain.](image1)

![Fig. 2: Typical finite element mesh used in current study in Lusas.](image2)

Material properties of steel for composite was used in finite element simulation are given in Table 1 (Anwar Hossain 2003; Soroush Amiriand Reza Masoudnia, 2010).

| Table 1: Steel material properties for the composite column in finite element simulation. |
|-----------------------------------------------|-----------------|
| Steel Properties                              | Values          |
| Density                                       | 7800 kg/m³      |
| Modulus of Elasticity, Es                     | 209×10³ N/mm²   |
| Poisson’s Ratio, ν                             | 0.3             |
| Yield Stress, fy                              | 275 N/mm²       |

The concrete material with Compressive strength 25 N/mm² and Density 2400kg/m³ used in this study.

Finite Element Discretization:
Verification is needed in order to ensure that the simulation process is correct. Short composites columns tested by Anwar Hossain were analyzed. This paper describes the behavior of short composite filled columns under axial compression and bending moment, the information summarized in Table 1, the dimensional properties of typical columns are indicated in Figure 3 where ‘a’ and ‘b’ represent cross-sectional dimensions with b representing minimum dimension of the columns. ‘h’ and ‘t’ represent height of the column and thickness of the steel plate, respectively (Anwar Hossain, 2003).
In general, residual stresses and boundary conditions play significant role in the structural behavior. In fact, it can be argued that an accurate finite element simulation may take longer to set up and execute than a carefully controlled test. Hence verification study becomes very important in calibrating and implementing the finite element analysis. Once this done, the finite element analysis can be used to extend the range of test data, and to investigate the effect of changing variables, such as stress-strain characteristics, residual stresses, geometric imperfections and section geometry (Tomiai Sakino, 1979; Furlong 1967; Gourley 2001; Schneider 1999; Liu and Kennedy, 1994; Knowles and Park, 1970; Sabnis, 1979; Young-Hwan Choi, 2004; Anwar Hossain, 2003; Liang, Brian Uy, J.Y. Richard Liew, 2004; Liang, Brian Uy, J.Y. Richard Liew, 2007).

The curves of load displacement of the columns are plotted against the experimental data in Figure 4. Generally, the numerical results show good agreement with experimental data.
The verification study showed that the finite element model is able to predict the behavior of thin-walled box composite columns with sufficient accuracy (Tomian and Sakino, 1979; Furlong, 1967; Gourley, 2001; Schneider, 1998; Lu and Kennedy, 1994; Knowles and Park, 1970; Sabnis, 1979; Young-Hwan Choi, 2004; Anwar Hossain 2003).

RESULTS AND DISCUSSIONS:

In view of the accuracy of the finite element model proposed, the method was used for the analysis of composite columns with same ends supports as those tested by Anwar Hossain in 2003 but with different size of cross-sections and lengths were analyzed and the effect of these changes on the ultimate load capacity and load deflection behavior of columns were studied.

Prismatic Study of Uniform Composite Columns Subjected to Axial load:

All short columns have the same dimensions at the two ends (600 mm), the thicknesses of steel tube was increased from 5 mm to 30 mm. The columns divided into four categories within 4 different lengths, 4 m, 8 m, 12 m and 16 m. Figure 5 shows the schematic diagram of the column. The columns were analyzed and the results obtained in terms of load-deflection relationship and ultimate load capacity in Figure 6.
Interaction Diagram for Uniform Composite Columns:

Use of beam-columns is common in building construction. Beam-columns are subjected to the combined action of axial load and bending moment. Columns of different proportions were also analyzed under such loading. The effect of the combined action of axial load and bending moment is obtained as interaction diagrams.

Exterior columns in space frames are normally subjected to axial load and bending moment. Figure 7 shows the interaction diagrams obtained for the short columns subjected to an only axial ultimate load in the axial direction in the absence of any moments, only ultimate bending moment about the axis of bending and varying bending moment with axial load applied. Curves are plotted for uniform columns by 360000 mm² in hinge end and roller end are with different thicknesses from 5 mm to 30 mm and different lengths from 4 m to 16 m. The figures indicate clearly the complementary nature of the resistance against bending about one axis in the presence of axial load.

These figures show that all columns behave in same manner with different values of lengths and thicknesses in long uniform columns.

The short columns in these cases were analyzed under axial load and bending moment. It is obvious from the figure that axial load capacity drops as the magnitude of the bending moment is increased and vice versa, but there is a limitation where the moment capacity will be increased as the axial load is increased. This phenomenon is happened in composite columns not in steel columns.

**Fig. 6:** Load-displacement plots for uniform cross sections.
Conclusions:
Verification of finite element analysis of the short composite box columns was carried out and compared with the experimental studies on the columns which based on the mentioned results it can be concluded that the proposed three dimensional finite elements modelling using the LUSAS software is sufficiently accurate in predicting the ultimate load capacity and behaviour of the composite box columns. The ultimate load capacity and the behaviour of the composite short box columns with different size shapes and cross-sections and lengths have been considered in this study by the use of finite element analysis and It is concluded that the thickness of tube plate play important role in failure modes and the ultimate load capacity of the pin ended short composite box columns.

Appendix: List of Abbreviation.

<table>
<thead>
<tr>
<th>Name of Column</th>
<th>Column Shape</th>
<th>Length of Column</th>
<th>Top Side width</th>
<th>bottom Side width</th>
<th>Thickness of Steel Plat</th>
</tr>
</thead>
<tbody>
<tr>
<td>UC-4-5</td>
<td>Uniform</td>
<td>4 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>5 mm</td>
</tr>
<tr>
<td>UC-4-10</td>
<td>Uniform</td>
<td>4 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>10 mm</td>
</tr>
<tr>
<td>UC-4-20</td>
<td>Uniform</td>
<td>4 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>20 mm</td>
</tr>
<tr>
<td>UC-4-30</td>
<td>Uniform</td>
<td>4 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>30 mm</td>
</tr>
<tr>
<td>UC-8-10</td>
<td>Uniform</td>
<td>8 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>10 mm</td>
</tr>
<tr>
<td>UC-8-20</td>
<td>Uniform</td>
<td>8 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>20 mm</td>
</tr>
<tr>
<td>UC-8-30</td>
<td>Uniform</td>
<td>8 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>30 mm</td>
</tr>
<tr>
<td>UC-12-20</td>
<td>Uniform</td>
<td>12 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>20 mm</td>
</tr>
<tr>
<td>UC-12-30</td>
<td>Uniform</td>
<td>12 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>30 mm</td>
</tr>
<tr>
<td>UC-16-30</td>
<td>Uniform</td>
<td>16 m</td>
<td>600 mm</td>
<td>600 mm</td>
<td>30 mm</td>
</tr>
</tbody>
</table>
REFERENCES


LUSAS help version 14.


YOUNG-HWAN CHOI, 2004. A modified AISC P-M Interaction Curve for Square Concrete Filled Tube Beam-Column, University of Illinois at Urbana-Champaign, USA