Jurnal
Teknologi

Full paper

Numerical Simulation of Cold-Formed Steel Top-Seat Flange Cleat Connection

Cher-Siang Tan[a*], Yeong-Huei Lee[a], Yee-Ling Lee[a], Shahrin Mohammad[a], Arizu Sulaiman[a], Mahmood Md Tahir[b], Poi-Ngian Shek[b]

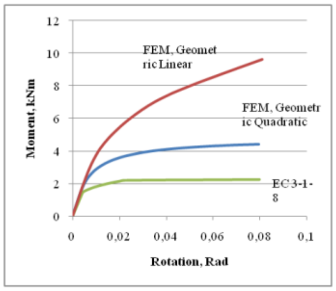
[a]*Faculty of Civil Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia*
[b]*Construction Research Centre, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia*

*Corresponding author: tcsiang@utm.my

Article history

Received :1 November 2012
Received in revised form :15 January 2013
Accepted :15 March 2013

Graphical abstract



Abstract

This paper presents the numerical simulation of cold-formed steel top-seat flange cleat connection in light steel framing. The cold-formed steel channel sections are assembled back-to-back to form I-shaped beam and column members. Angle sections are used to connect the beam to column with the profile of top-seat flange cleat connection. Numerical models are developed to predict the moment-rotation behaviour of such joint profile. Verification of numerical model with Eurocode 3 is carried out for solving mathematical problems, whereas validation with experimental results is done to investigate the physical uncertainties such as errors removal and uncertainties evaluation. Convergence study is carried out from the perspectives of geometry and meshing in order to optimise the developed finite element models. The result indicated that finite element analysis is suitable for the structural analysis on cold-formed steel top-seat flange cleat connection.

Keywords: Numerical simulation; finite element analysis; top-seat flange cleat connection; cold-formed steel; light steel framing

Abstrak

Kertas kerja ini membentangkan kerja simulasi berangka bagi keluli tergelek sejuk dengan sambungan atas-bawah cleat bebibir dalam rangka keluli ringan. Keratan "channel" keluli tergelek sejuk dipasang berkembar untuk membentuk anggota rasuk dan tiang berbentuk-I. Keratan "angle" digunakan untuk menyambung rasuk dan tiang dalam bentuk "atas-bawah cleat berbibir". Model unsur terhingga dibangunkan untuk meramalkan kelakuan momen-putaran sambugan ini. Pengesahan model berangka dengan model Eurocode 3 dijalankan bagi menyelesaikan masalah matematik, manakala validasi dengan keputusan eksperimen dijalankan untuk menyiasat isu yang berkiatan dengan fizik seperti penyingkiran kesilapan dan penilaian ketidaktentuan. Kajian penumpuan dijalankan dari perspektif geometri dan jaringan dalam usaha untuk meningkatkan kebolehpercayaan model unsur terhingga yang dibangunkan untuk mewakili kelakuan sebenar sambungan. Keputusan penyelidikan ini menunjukkan bahawa analisis unsur terhingga adalah sesuai digunakan untuk analisa struktur terhadap keluli tergelek sejuk dengan sambungan atas-bawah cleat berbibir.

Kata kunci: Penyelakuan berangka; analisis unsur terhingga; sambungan atas-bawah cleat bebibir; keluli tergelek sejuk; rangka kekuli ringan

■1.0 INTRODUCTION

Light steel framing with cold-formed steel sections is one of the effective solutions for Industrial Building System (IBS). Cold-formed steel has been actively applied in construction industry due to its high strength-to-weight ratio, compared to identical hot-rolled steel section. Besides that, as a reusable material, it also contributes to the construction of green and sustainable building by minimise construction waste.

Joints in steel framing are important structural elements in transferring loads and moments. Commonly, joints consume the major proportion of the fabrication cost in steelwork construction. Simple and strong joints are favourable among engineering design for cost efficiency. The top-seat flange cleat joint that able to achieve semi-rigid has the potential to serve the cost-effective purpose. This joint profile offers simple configuration in fabrication and installation, and requires no welding. Mechanical model of top-seat flange cleat connection was developed by Raffaele (2001) [1] and was firstly included in the code of practice, Eurocode 3-1-8 (hereby addressed as EC3-1-8) [2] for hot-rolled steel design.

The researches on joint behaviour for cold-formed sections [3-8] show little contribution and has not been studied in-depth. As cold-formed steel tends to buckle, it is recommended to eliminate rigid connection option in order to prevent sudden failure and premature failure of the framing. However, the different structural performance may cause inadequacy when adopting current hot-rolled design procedures into cold-formed design.

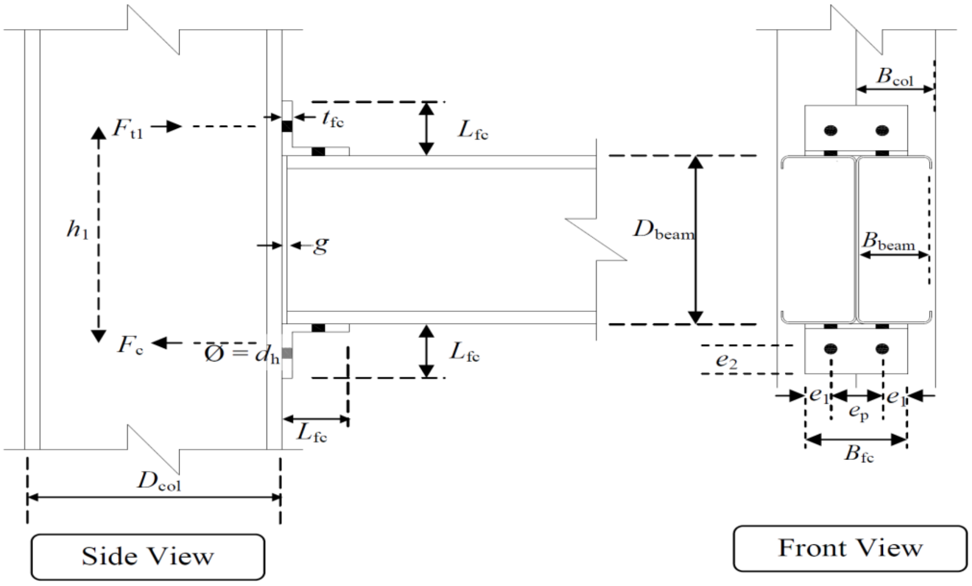
Since past few decades, finite element analysis is one of the effective ways to predict the structural behaviour of building. Citipitioglu et. al. [9] carried a study on 3D models of bolted connection with angles. Contact between components was modelled which included the effect of friction. Gantes and Lemonis [10] presented finite element model for bolted T-stub connections. Validation with experimental results was done by implementing material and geometrical nonlinearities.

This study is conducted to verify the modelling technique and validate with experimental results for cold-formed steel top-seat flange cleat connection in light steel framing. The finite element model is developed to ensure the convergence in finite element analysis is achieved and used to represent the actual behaviour of full-scale test.

■2.0 MODEL VERIFICATION

2.1 Eurocode 3 Model

Top-seat flange cleat connection, as shown in Figure 1, is analysed in accordance to the mechanical model given in EC3-1-8 [2]. The design resistance of the joint is calculated based on component method. EC3-1-8 [2] has provided the procedures to obtain the component resistances for further design purpose. Table 1 shows the dimension details of the joint in this study. The components resistance include bolt tension resistance, bending resistance of flange cleat, bending resistance of column flange, transverse tension resistance of column web, compression resistance of beam and column flange, compression resistance of beam and column web as well as shear resistance. Taking joint T2 as an example, Table 2 shows the tension resistances of the joint, which are calculated based on requirements shown in Table 6.1 of EC3-1-8 [2]. The bending resistance of column flange is the lowest resistance and thus is taken to calculate the moment resistance of the joint. A moment-rotation curve is plotted with the calculated joint moment resistance. For conservative design, Eurocode has limited the model behaves as elasto-plastic with no strain hardening characteristic.



Legend:

D_{beam} – depth of beam section
D_{col} – depth of column section
L_{fc} – length of flange-cleat
t_{fc} – thickness of flange -cleat
e_1, e_2 – edge distance from centre of bolt hole
e_p – distance between centre of bolt hole
d_h – diameter of bolt hole
g - gap between beam to column flange
B_{beam} – width of beam
B_{col} – width of column
B_{fc} – width of flange -cleat

Figure 1 Top-seat flange cleat connection

Table 1 Dimension of the studied top-seat flange cleat connections

Label	**Column Dimension, mm**	**Beam Dimension, mm**	**Flange Cleat Dimension, mm**	**Bolt**
T1	D_{col} = 250 B_{col} = 77 t_{col} = 2	D_{beam} = 150 B_{beam} = 66 t_{beam} = 2	L_{fc} = 100 B_{fc} = 60 t_{fc} = 6	M8.8 12mm bolt $e_1 = e_2$ = 25 mm e_p = 50 mm d_h =13 mm g = 2 mm
T2		D_{beam} = 200 B_{beam} = 72 t_{beam} = 2		
T3		D_{beam} = 250 B_{beam} = 77 t_{beam} = 2		

Table 2 Tension resistances of top-seat flange cleat connection calculated based on Table 6.1, EC3-1-8 [2]

Component	**Resistance, kN**
Tension resistance of bolt	97.1
Bending resistance of flange cleat	19.7
Bending resistance of column flange	8.8
Transverse tension of column web	77.3

2.2 Finite Element Model

Finite element models are developed to predict the structural behaviour of cold-formed steel top-seat flange cleat connection. Since it is costly to conduct full-scale isolated joint tests with various dimensions, finite element analysis is an effective alternative to predict the nodal moment and rotational stiffness in light steel framing.

According to ABAQUS benchmark and NAFEMS standard [11], for beam bending, static solution with solid or shell elements is suitable used in this finite element analysis for structural connection.

2.2.1 Element option

Shell or solid element is suitable to be applied into this finite element analysis. In simulation of a simple flexural beam using continuum or shell elements, it is found that they are too stiff in their flexural behaviour. This is due to a disproportionately large shear-related strain energy arises, which greatly increases the flexural rigidity of the model [11]. According to benchmark test [11], there are several alternative continuum element that can overcome this shear-locking deficiency.

With reference to the suggested solution, it was found that the second-order isoparametric elements are useful in the analysis. The element type of C3D20R, 20-node quadratic brick meshing was applied to the model. Brick element has showed the capability to represent large deformations, geometric and material nonlinearities [12].

2.2.2 ABAQUS/Standard

The suitability of Standard or Explicit analysis is always queried by user in structural analysis. In this particular problem, based on previous study [12], Standard analysis is applied in analysis. ABAQUS finite element analysis consists of two analysis methods, namely full Newton and Quasi-Newton. Newton's numerical technique is the common approach that requires Jacobian evaluation for each iteration in the system. The convergence rate from Newton method has the motivation in utilizing it into ABAQUS/Standard as compared to other alternatives with slower convergence rate.

In classical Newton method, the possibility to formulate Jacobian matrix in complex algorithm is low and expensive for each iteration. Full Newton method can be one of the alternatives by recalculating Jacobian matrix occasionally (as in the initial strain method of simple contained plasticity problems) [13].

Full Newton analysis and Quasi-Newton method are capable to solve nonlinear analysis. The decision of using full Newton or Quasi-Newton method is relied on the system symmetricity. Quasi-Newton is suitable for small-displacement analysis. Since the cold-formed connection experienced a large deformation, full Newton with extra convergence properties is suitable for finite element analysis to prevent from divergence problem.

2.2.3 Geometric Analysis

In most of finite element codes, stiffness method or displacement method is used in the solution.

$$Ku = f \qquad (1)$$

where K is stiffness, u is displacement and f is force. Hence, the fundamental finite element solution is nodal displacement. The stress and strain are calculated from these nodal displacement. Geometric analysis has direct influence in nodal displacement calculation. Suitable geometric should be assigned to the finite element model in order to achieve good agreement with experimental data.

In ABAQUS/Standard analysis, geometric is divided into linear and quadratic analysis. The analysis for different geometric analysis may give various results. Linear geometric uses a shorter time to complete the analysis but may inadequate for cold-formed steel sections which experienced large deformation.

2.2.4 Contact Behaviour

For interaction properties, the penalty friction formula and 0.31 friction coefficient was applied to the tangential behaviour. Difference in friction coefficient will not give significant impact to the model [14]. Meanwhile, in normal behaviour, penalty constraint enforcement method and "hard" contact were used [14].

The contacts may include bolt head with flange cleat, bolt thread with beam, bolt thread with flange cleat, flange cleat with beam, flange cleat with column, nut with column, nut with beam and bolt thread with column.

Surface-to-surface interaction was used for each contact pair. In this contact conditions, the enforcement is applied averagely over slave surface which may prevent some large and undetected penetration of master nodes into the slave surface in the analysis.

There are two contact tracking approach in the ABAQUS finite element analysis, namely finite sliding and small sliding tracking approach. Finite sliding contact is generally applied in the analysis and allows for arbitrary relative separation, sliding and rotation. Nevertheless, small sliding contact assumes a little sliding based on linearized approximations of the master surface per constraint [13]. Small sliding option was chosen in the analysis to fully transfer loading to the supporting member.

Penalty method has the significant contribution in reducing solver cost (in terms of CPU time and memory) by not generate additional degrees of freedom [13]. Moreover, the numerical softening associated with the penalty method can improve the robustness to resolve difficult contact situations.

2.2.5 Material Properties

The model was divided into four components, namely beam, column, angle and bolt. The thickness of the beam and column sections is 2 mm while the thickness of flange cleat is 6 mm. The components are assigned as elastic-perfectly-plastic behaviour with yield strength of 350 N/mm^2. In ABAQUS finite element analysis, elastic zone is analyzed by Poisson ratio and Young's modulus.

2.2.6 Loading and Boundary Conditions

The loading was applied at a distance of 1000 mm from column surface. The boundary conditions were set such that the base of the column was fixed, top of column was retrained in two direction to prevent the structure to sway and beam was restrained to prevent in lateral-torsional buckling.

2.2.7 Meshing

Meshing is one of the important elements in finite element analysis. The meshing of these models may give significant influence to the results obtained. Coarse mesh would cause numerical convergence problem and not able to show buckling characteristic for thin-walled behaviour of cold-formed section [14]. Meanwhile, smaller mesh would extend the computation time [14].

The model is well-partitioned before meshing applied to the model. Edge cutting technique was used in the partition process. All elements in the models are structured hex meshing.

2.3 Benchmarking Developed Model

Eurocode 3 mathematical model [2] as discussed in the previous section is selected as a benchmark for cold-formed steel top-seat flange cleat connection. The stiffness of the joint is determined by moment-rotation characteristic, where,

if $M_{j,Ed} < 2/3\ M_{j,Rd}$ (2)

$$\phi = \frac{M_{j,Ed}}{S_{j,ini}}$$

if $2/3\ M_{j,Rd} < M_{j,Ed} < M_{j,Rd}$ (3)

$$\phi = M_{j,Ed} \times \frac{\left(\dfrac{1.5M_{j,Ed}}{M_{j,Rd}}\right)^{\varphi}}{S_{j,ini}}$$

$M_{j,Rd}$ is design moment resistance of the connection, $M_{j,Ed}$ is moment applied, Ψ is 3.1 for bolted angle flange cleat and $S_{j,ini}$ is initial stiffness of connection.

The following formulas are to determine the moment capacity of connection, M_j in top-seat flange cleat joint, where,

$$h_1 = D_{beam} + L_{fc} / 2 + (L_{fc} - e) \quad (4)$$

$$M_j = \sum F_t\, h_1 \quad (5)$$

where F_t is the total tension resistance, h_1 is the distance of bolt row 1 to centre of compression load, D_{beam} is the depth of beam, L_{fc} is the length of leg in flange cleat and e is the edge distance.

2.4 Verification Of Numerical Model

Verification of model is supposed to deliver evidence that mathematical models are properly implemented and that the numerical solution is correct with respect to the benchmarked mathematic model. The high von Mises stress may contribute to the failure at low stiffness component. Figure 2 shows the von Mises stress distribution for developed model. The high stress areas are included bolt, nut, flange cleat and column flange. Table 3 shows the calculated stiffness of EC 3 model. The failure is suspicious to occurred at the flange of column.

Table 3 Stiffness coefficients calculated for EC3 model based on Table 6.1 of EC3-1-8 [2]

Component	Stiffness coefficient, *k*
Column web panel in shear	1.657
Column web in compression	0.253
Column web in tension	0.708
Column flange in bending	0.115
Flange cleat in bending	0.297
Bolts in tension	3.645
Bolts in shear	2.194
Bolts in bearing	1.646

Figure 3 shows the moment-rotation behavior of top-seat flange cleat connection based on T2 beam's profile, and the yield stress (f_y) is fixed at 350 N/mm^2. Referring to the figure, finite element models have showed a slightly higher elastic stiffness as compared to EC 3 model. The significant different is showed on the plastic region. Geometric in quadratic formulation can be used to represent the large deformation in the analysis as compared to linear geometric. Metal ductility may contribute to the geometric nonlinearity in deformation of the analyzed elements. Furthermore, thin-walled behaviour of cold-formed steel structures may not follow the linear geometric analysis due to lower stiffness or experience lateral instability. In order to fulfil the actual metal behaviour, geometric quadratic analysis is applied in the finite element models which has better agreement with EC 3 benchmark model.

There is still a significant gap between EC 3 model and finite element model in the plastic region. This is due to EC3 model assumes failure at the first component and does not associate with other component. Finite element model involves other components in the resistance after the failure of the first component, thus contributes to higher plastic resistance behaviour. As the connection is developed by several components, it is essential to include the interaction between the components. The finite element model can be accepted for further validation process.

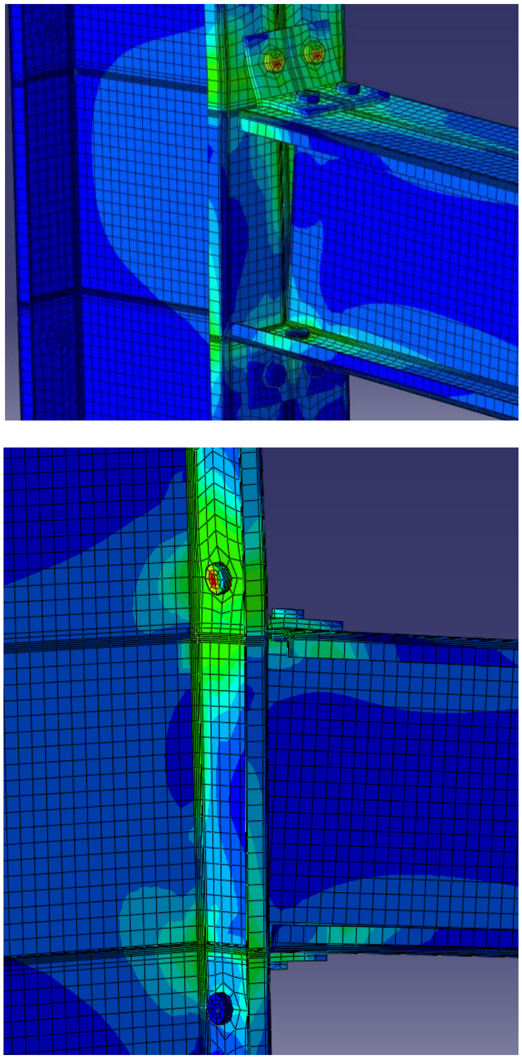


Figure 2 Von Mises stress distribution for developed numerical model

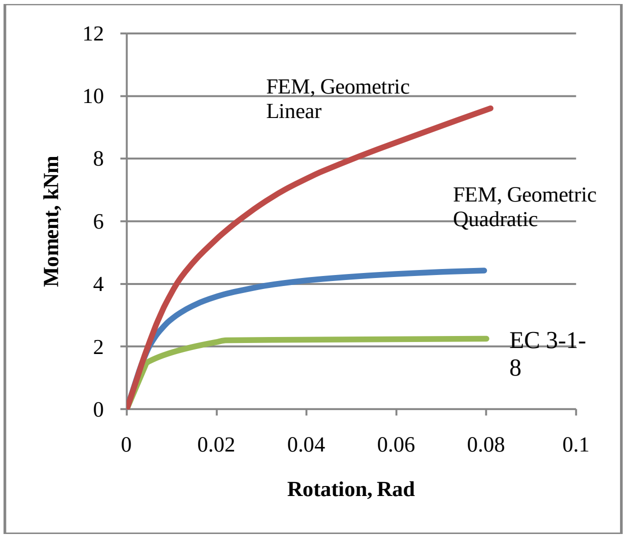


Figure 3 Moment-rotation curves based on mathematical model and numerical models

■3.0 MODEL SIMPLIFICATION

3.1 Geometrical Symmetry Model

Since the geometry of the model is symmetry, the model is simplified to a half model in order to save the analysis time. The simplification is made because there is no unequal action acted on the model. Figure 4 shows half and full finite element models in the analysis. The model is pinned at symmetrical line to prevent from distortional buckling.

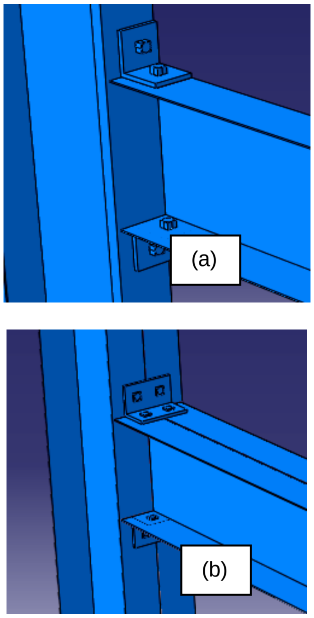


Figure 4 (a) Half model and (b) full model for numerical investigation

3.2 Mesh Convergence

The meshing convergence study is important to identify the optimum meshing size in this study. The work is carried out base on geometric with linear formulation and quadratic formulation, applied to suitable mesh size for further analysis. The study is carried out on a specific node on beam component in order to obtain its von Mises stress.

Figure 5 shows the mesh convergence study based on linear formulation. Models with mesh size of 100 mm is analyzed, followed by smaller mesh size of 50 mm, 40 mm, 30 mm, 20 mm, 10 mm, 5 mm until 1 mm. The convergence study shows an optimized mesh start from 4 mm for linear formulation. A size of 10 mm mesh size of geometrical quadratic formulation achieves a similar stress with 4 mm mesh size of geometrical linear formulation. Hence, the 10 mm mesh size with geometrical quadratic formulation is applied in the finite element model.

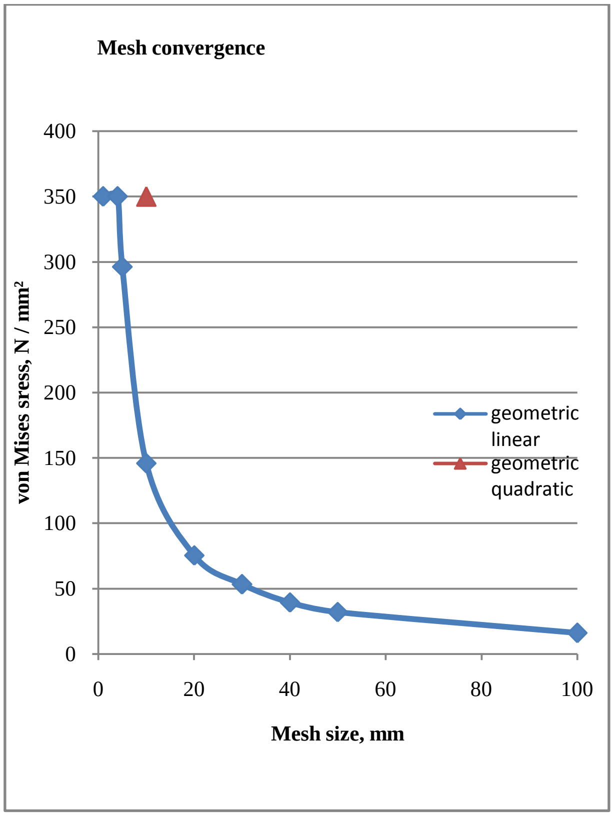


Figure 5 Mesh convergence study

4.0 MODEL VALIDATION

4.1 Experimental Investigation

A series of three experimental investigation (T1 to T3) was carried out at Universiti Teknologi Malaysia. The moment-rotation behaviour was recorded from the experimental study [15]. Figure 6 shows the experimental model of top-seat flange cleat joints with the dimensions of the connection are described in Table 2. The beams were placed at a 2 mm gap from the column flange and 1.5 m from the top of the column. The length of the beam was 1.5 m and the length of column was 3 m. The bottom of the column was restrained in all directions, while the top of the column was prevented from moving in the horizontal direction. A point load was applied to the beam, at a distance of 1 m from the column flange. The same length of flange cleat and same size of bolt were used to accommodate the limited space between connected members of the flange, as well as to satisfy the minimum end bearing distance. The data acquisition system consisted of inclinometers and distance transducers. Tensile tests were conducted to determine the material properties of the cold-formed steel [15]. The results are inserted into finite element model to mimic the actual material behaviour.

Figure 6 Experimental model of the top-seat flange cleat connection

4.2 Validation of Numerical Model

The moment-rotation curve is a good characteristic relationship to represent the stiffness and strength behaviour of top-seat flange cleat connection. From the Figure 7 to Figure 9, finite element model (FEM) shows a closed comparison with experimental results (EXP). The percentage of difference for the initial stiffness of joints ($S_{J,ini}$) between experimental and numerical model ranged up to 7%. The moment resistance of joints taken at 0.03 Rad showed up to 9% difference between experimental and numerical results. Both comparisons showed that the model achieved a good agreement with the experimental results.

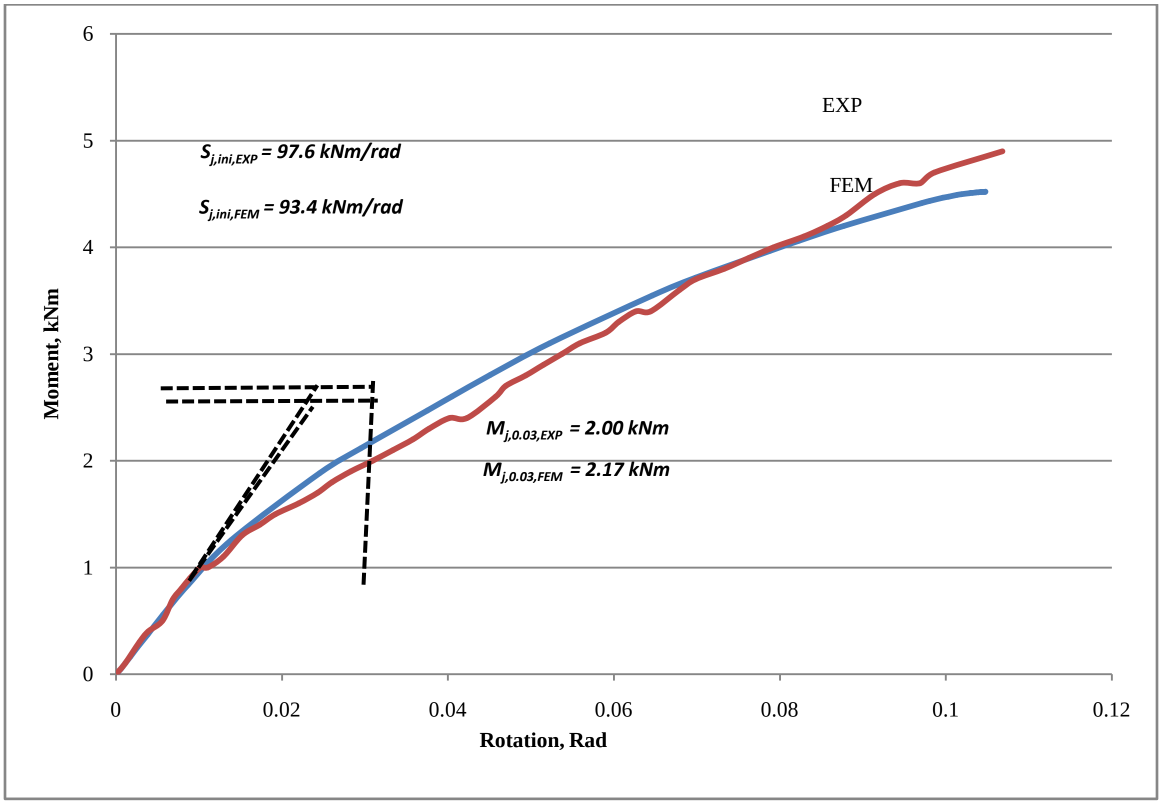


Figure 7 Moment-rotation behaviour for model validation for T1

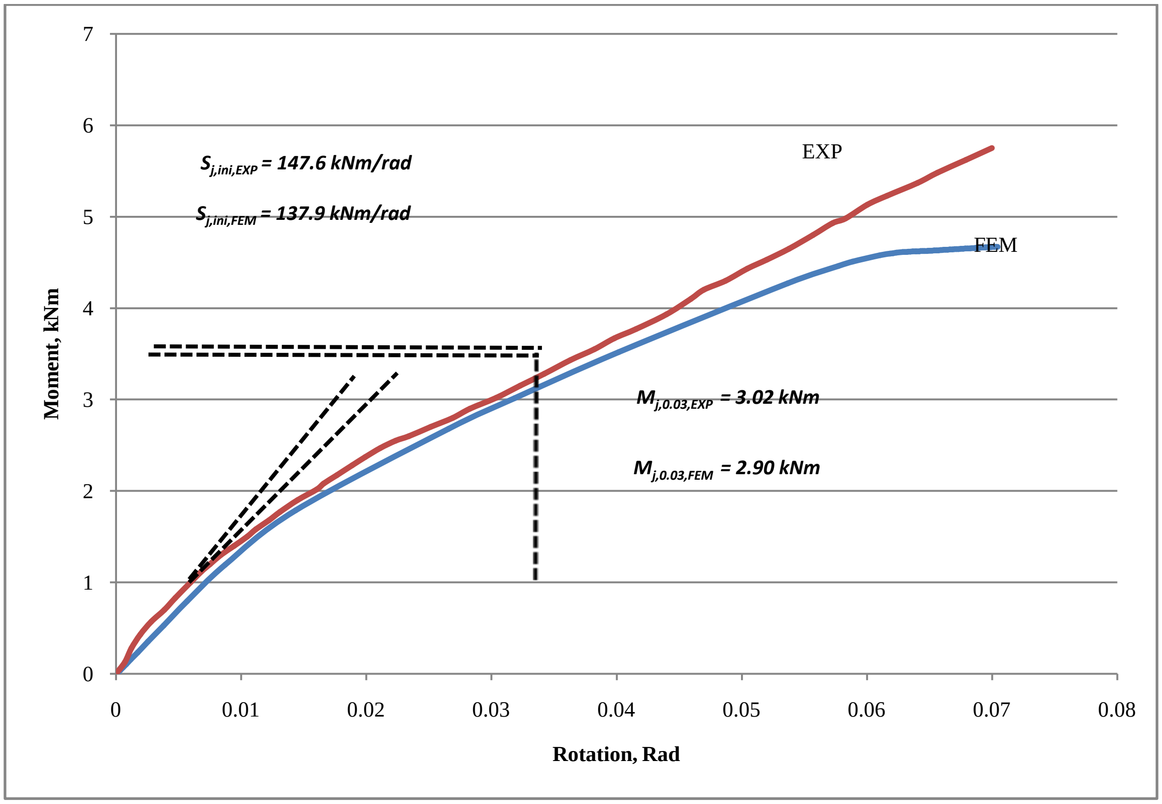


Figure 8 Moment-rotation behaviour for model validation for T2

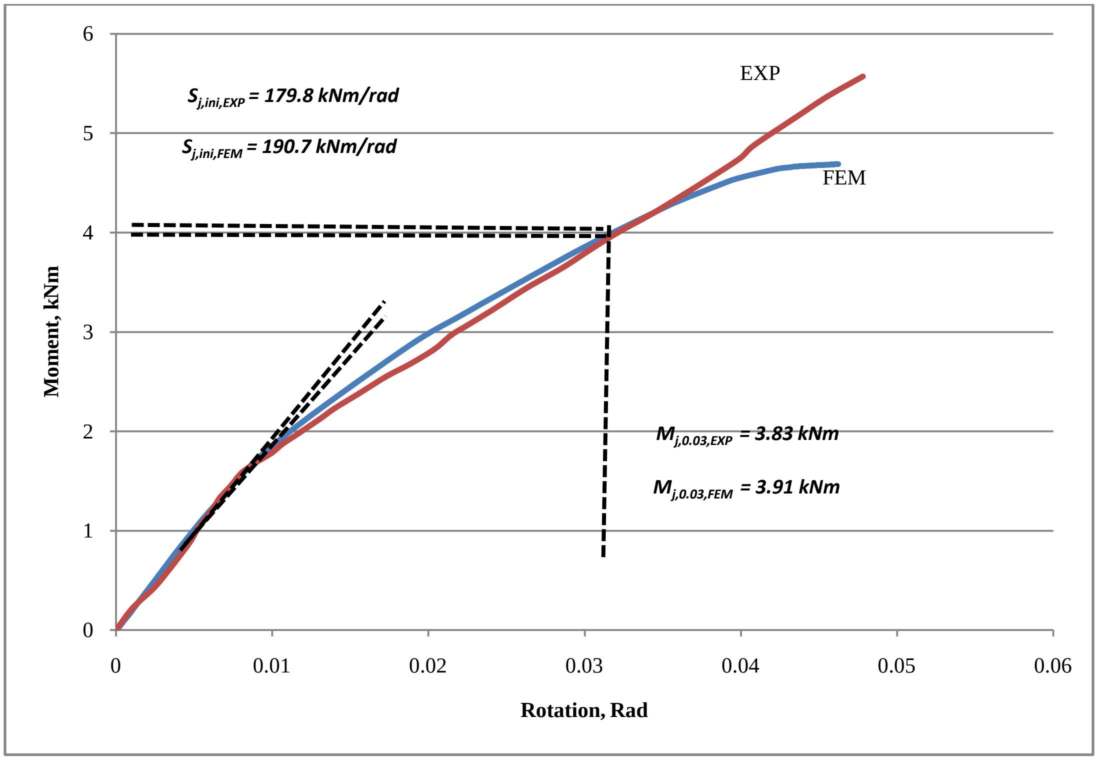


Figure 9 Moment-rotation behaviour for model validation for T3

From Figure 10, the failure mode for experimental investigation and numerical study has shown the similarity. Both deformation patterns show the same failure at bended column flange. The flange cleat is not the critical part in the structural behaviour due to greater resistance with thicker cross-section.

In order to model 6 mm flange cleat connection, nonlinearity is recommended in the analysis. ABAQUS/Standard analysis with static solver, geometrical quadratic formulation, material nonlinearities, boundary well-defined and sufficient meshing size can be applied in cold-formed steel top-seat flange cleat connection.

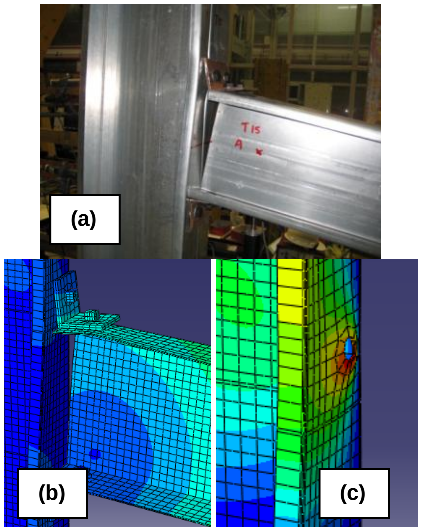


Figure 10 Failure modes: (a) experimental investigation (b) FEM investigation (c) detailed failure at the column's flange from FEM

5.0 CONCLUSIONS

From the investigation, finite element modelling (FEM) technique using ABAQUS/Standard can be concluded. ABAQUS/Standard analysis with static solver, material nonlinearity, half simulation, geometric nonlinearity, convergence mesh size of 4 mm for linear formulation and 10 mm mesh size for quadratic formulation, well-defined boundary condition and loading are used for the top-seat flange cleat connection in light steel framing. Nonlinearity has showed its importance in this study due to the cold-formed steel joints tend to rotate at large-deformation.

The FEM models show similar failure modes as compared to physical experimental testing results. The initial stiffness and moment resistance of joints are differed up to 7% and 9% respectively in comparison between FEM and experiment results.

The result obtained from this study is limited to the connection profiles and their respective dimension. It is suggested to carry out further parametric study with the same technique for finite element modelling to achieve better confidence for such structural joint's analysis and design.

Acknowledgement

This reported research was funded by a research grant from Ministry of Higher Education, Malaysia (MOHE), Universiti Teknologi Malaysia (research grant 05J76) and ICON Research Alliance (UTM-ICON). The technical and financial supports are gratefully acknowledged.

References

[1] Raffaele, P. 2001. Top-and-seat and Web Angle Connections: Prediction Via Mechanical Model. *Journal of Constructional Steel Research*. 57: 661–694.

[2] British Standards Institution. 2005. Eurocode 3: Design of steel structures: Part 1.8: *Design of Joint*. BS EN 1993-1-8. U.K. BSI.

[3] Kim, T. S. and H. Kuwamura. 2007. Finite Element Modeling of Bolted Connections In Thin-Walled Stainless Steel Plates Under Static Shear. *Thin-walled Structures*. 45: 407–421.

[4] Kim, T. S., H. Kuwamura and T. J. Cho. 2008. A Parametric Study on Ultimate Strength of Single Shear Bolted Connection With Curling. *Thin-walled Structures*. 46: 38–53.

[5] Kim, T. S., H. Kuwamura, S. H. Kim, Y. T. Lee and T. J. Cho. 2009. Investigation on Ultimate Strength of Thin-Walled Steel Single Shear Bolted Connections with Two Bolts Using Finite Element Analysis. *Thin-walled Structures*. 47: 1191–1202.

[6] Wong, M. F. and K. F. Chung. 2002. Structural Behaviour of Bolted Moment Connections in Cold-formed Steel Beam-column sub-frames. *Journal of Constructional Steel Research*. 58: 253–274.

[7] Lim, J. B. P. and D. A. Nethercot. 2003. Ultimate Strength of Bolted Moment-connections between Cold-formed Steel Members. *Thin-walled Structures*. 41: 1019–1039.

[8] Lim, J. B. P. and D. A. Nethercot. 2004. Stiffness Prediction for Bolted Moment-connections between Cold-formed Steel Members. *Journal of Constructional Steel Research*. 60:. 85–107.

[9] Citipitioglu A. M., R. M. Haj-Ali and D. W. White. 2002. Refined 3D Finite Element Modeling of Partially-Restrained Connection Including Slip. *Journal of Constructional Steel Research*. 58: 995–1013.

[10] Gantes C. J. and M. E. Lemonis. 2003. Influence of Equivalent Bolt Length in Finite Element Modeling of T-stub Steel Connection. *Computer & Structures*. 81: 595–604.

[11] Dassault Systèmes. *ABAQUS Benchmarks Manual*. 2009. United States of America. ABAQUS Inc.

[12] Yang, B. and K. H. Tan. 2012. Numerical analyses of steel beam-to-column joints subjected to catenary action. *Journal of Constructional Steel Research*. 70: 1–11.

[13] Dassault Systèmes. *ABAQUS Analysis user's manual version 6.9*. 2009. United States of America. ABAQUS Inc.

[14] Dai, X. H., Y. C. Wang, and C. G. Bailey. 2010. Numerical Modelling of Structural Fire Behaviour of Restrained Steel Beam-column Assemblies using Typical Joint Types. *Engineering Structures*. 32: 2337–2351.

[15] Tan, C. S. 2009. *Behaviour of Pin and Partial Strength Beam-to-Column Connections with Double Channel Cold-Formed Steel Sections*. PhD thesis, Universiti Teknologi Malaysia.