Nanotechnology Perceptions
ISSN 1660-6795
www.nano-ntp.com

Numerical Approach of Modified T-Stub Steel Connection

Totok Andi Prasetyo, Sumirin, Antonius

Student of Civil Engineering Doctoral Program, Postgraduate Faculty
Sultan Agung Islamic University, Indonesia
Email: totokandi77@yahoo.com

This research presents a three-dimensional numerical modeling process for the investigation of the Modified T – Stub Steel Connection. This research examines the connection behavior using a rigorous use of Idea Statistical and the finite element computation program ANSYS 19.2. Steel connection's nonlinear 3D continuum elements.
Keywords: Steel Connection with Modified T – Stubs Finite Element, non – Linear, Connection behavior.

1. Introduction

The subject of three-dimensional finite element modeling of end connections is covered in a number of other articles. A three-dimensional finite element model of Modified T – Stub connections has been created using the ANSYS finite element program. Specifically, the paper provided a prediction model to analyze the Modified T Stub Steel Connection, tracking the sources of rotation for the end-plate. The modeling method used to evaluate T Stub Steel Connection's ductility needs. A non-linier solution incorporating material non-linearities, contact surfaces, and large deflection analysis was reported for a single T Stub Steel joint using an ANSYS finite element model with solid elements. The modified connections computational study has gone successfully.

This study uses 3D modeling to identify contact effects and bolt geometry in Modified T-Stub Connections. Since the contact surface, bolts, and non-linearities of the material are the most crucial parameters, the stiffener and connection need to be modeled as accurately as possible. For the main components of the connection, such as the beam, column, and T Stub connection, the connection model is thought to primarily use shell elements.

In this study, the modified T – Stub Steel Connection is examined. After presenting the finite element of these connections, the behavior of the connection with the T – Stub Connection is finally anticipated.

Objective of the Study

a. To determine the performances and behavior Modified T Stub Steel Connection.

b. To determine the maximum stress of material.

Scope and Limitation

a. The model 1 column WF 400 x 200 x 8 x13 beam WF 300 x 150 x 6.5 x 9

b. The model 2 column WF 588x300x12x20 beam WF 450x200x9x14

c. The model 3 column WF 588x300x12x20 beam WF 500x200x10x16

d. Material Properties Fy = 240 MPa Fu = 400 MPa Bolt A325

e. Numerical analyzed with the ANSYS 19.2 and Idea Statistical programs.

Significance of the Study

a. The use of Modified T Stub Steel connection can be used as Special Moment Frame Prequalification connection.

b. For Installing on site are easier and faster and can reduce time schedule

Numerical Method with ANSYS 19.2 and Idea Statistical Program

The finite element method is a numerical analysis used in solving technical problem such as differential and integral equations with the approach method.

Tension Capacity for Bolt Tension Resistance Check (AISC 360-16: J3-1)

$\phi Rn = \phi \,.\, Fnt \,.\, Ab$

where:

Fnt = Nominal tensile stress

= 620 MPa (AISC 360-16 Table J3.2)

Ab = Gross bolt cross-sectional area

Ø = Resistance factor (LRFD)

= 0,75

$\phi Rn \geq Ft$

Shear Resistance Check (AISC 360-16: J3-2)

where:

$\phi Rn = \phi \,.\, Fnv \,.\, Ab$

Fnv = Nominal shear stress

= 372 MPa (AISC 360-16 Table J3.2)

Ab = Gross bolt cross-sectional area

Ø = 0.75 Resistance factor (LRFD)

Bearing Resistance Check (AISC 360-16: J3-2)

Where: Rn = 1,2. $lc. t . Fu \leq$ 2,4. $d. t . Fu$

lc = Clear distance, in the direction of the force, between the edge of the hole and the edge of the adjacent hole or edge of the material

t = Thickness of the plate

d = Diameter of a bolt

Fu = Tensile strength of the connected material

= 400 MPa

Ø = Resistance factor for bearing at bolt holes

= 0,75 (LRFD)

Check Capacity for Stiffening Plate

$\phi Rn = \phi . Fnw . Awe$

Fnw = nominal stress of weld material

Fnw = 0,6. $Fexx$ (1+0,5. sin□, □ θ)

Fexx = Electrode classification umber, i.e minimum specified tensile strength

Ø = Angle of loading measured from the weld longitudinal axis

Awe = Effective area of weld critical element

Ø = Resistance factor for welded connections

Finite Element Modelling

The modified T Stub Steel Connection was numerically modeled using the ANSYS finite element tool. Simple multi-linear stress-strain curves were employed for the bolt shanks and steel sections. The rate of convergence for a sub-steo of loading following yielding or through excessive component deformation was used to characterize the model's failure. The Modified T Stub's changing area of contact between its faces results in geometric non-linearities in addition to material non-linearities.

Element Type

Numerical modeling analysis is performed using the general-purpose finite element program ANSYS version 19.2. The Modified T – Stub Steel Connection uses the following ANSYS element classes. To represent beams, columns, and plates, 3D 4-node plastic small strain shell elements, or SHELL143, are utilized. With the aid of eight-node isoperimetric solid elements, or SOLID185, the bolt head, shank, and nut are idealized. Element CONTA178, a 3D node-to-node contact element, simulates the interactions between the plate and column. The model spar and bolt preload are handled by the LINK8 element. The Multi-linier stress-strain curves depict the material behavior for each element.

Model Configuration

Modified T – Stub that is nonlinear the steel connection is symmetrical with respect to the column web's center. A solid hexagon element is used to simulated the bolt head and nut. The spar element, which joins the head and nut nodes at the farthest corners, is used to simulate the bolt shank. The spar elements share one-twelfth of the bolt's effective area evenly. The bolt holes have a round model. Applying equal first strains on bolt shank elements simulates bolt pretension brought on by tightening the bolt. The bolt shares nodes with the plate ones because, after tightened, the head and/or nut remain in close proximity to their connecting angles and flanges.

In order to account for the impact of contact between the column flange and T Stub, an interface element is used. The model of the contact element is displayed in figure below. An initial gap of a three-dimensional point-to-point contact element is the name given to the interface element model. The interface elements link the nodes behind the angles to matching nodes at the web, column, and beam flanges. The maximum anticipated force divided by the maximum permitted surface displacement yields the normal stiffness value and sticking stiffness value.

To avoid the contact nodes from penetrating too deeply, a high contact stiffness was required. The mild carbon steel material parameters for the FE model are ascertained using the trilinear elastic-plastic technique. The yield stress is defined as 0.2% proof stress, and the material parameters of high strength steel are determined for the FE model using a multilinear elastic-plastic technique.

Applying initial strain simulates the bolt pretension that results from tightening the bolt. The purpose of the interface element is to handle the influence of praying forces. The nodes at the back of the plate are connected to matching nodes at the column flange via the interface components. To avoid the contact nodes from penetrating too deeply, a high contact stiffness was required. It is crucial to offer a bolt model that, while utilizing few components in its building, fulfills the total deformation. The employed finite element model is shown figure below.

Finite Element Result

Steel material properties for FE analysis is based on experimental data. All steel materials for connection used are A36 and M20 bolts grade A325. as shown in Table 1.

Table 1: Material properties for FE model

Specimen	Steel Grade	$\square_y$ (N/mm^2)	$\square_u$ (N/mm^2)
Column	A36	240	400
Beam	A36	240	400
Bolt	Grade A325	420	620
T Stub plate	A36	240	400

Figure behavior Modified T Stub model 1 shown moment and rotation drift angle more than 0.06 radian, column size WF 400x200x8x13, beam size WF 300 x 150 x6.5x 9

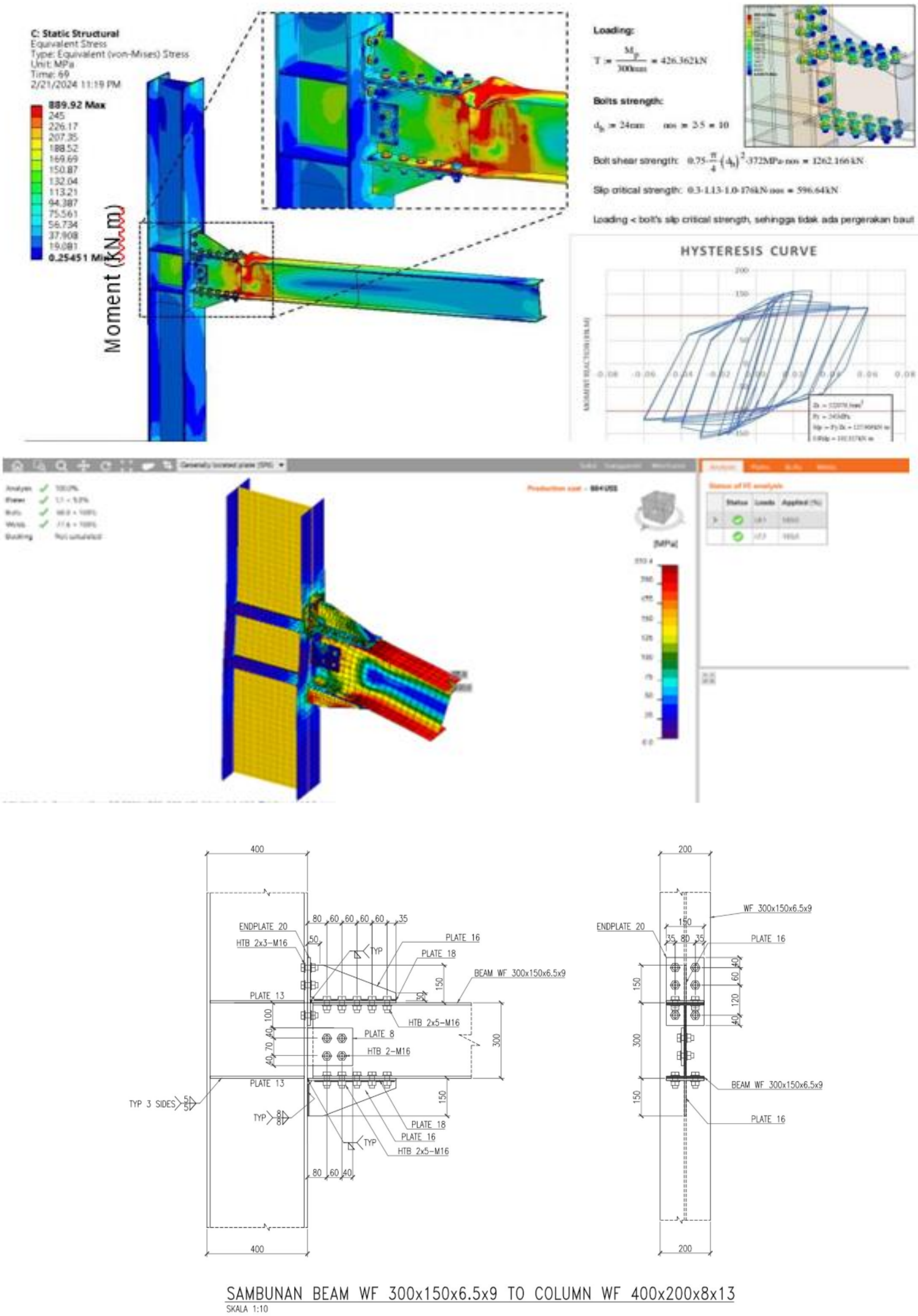

Figure 1: Behaviour hysteresis curve of model 1 Ansys and Idea Statistical

Figure 2 behavior Modified T Stub model 2 shown moment and rotation drift angle more than 0.06 radian, column size WF 588 x 300 x 12 x20, beam size WF 450 x 200 x 9 x 14

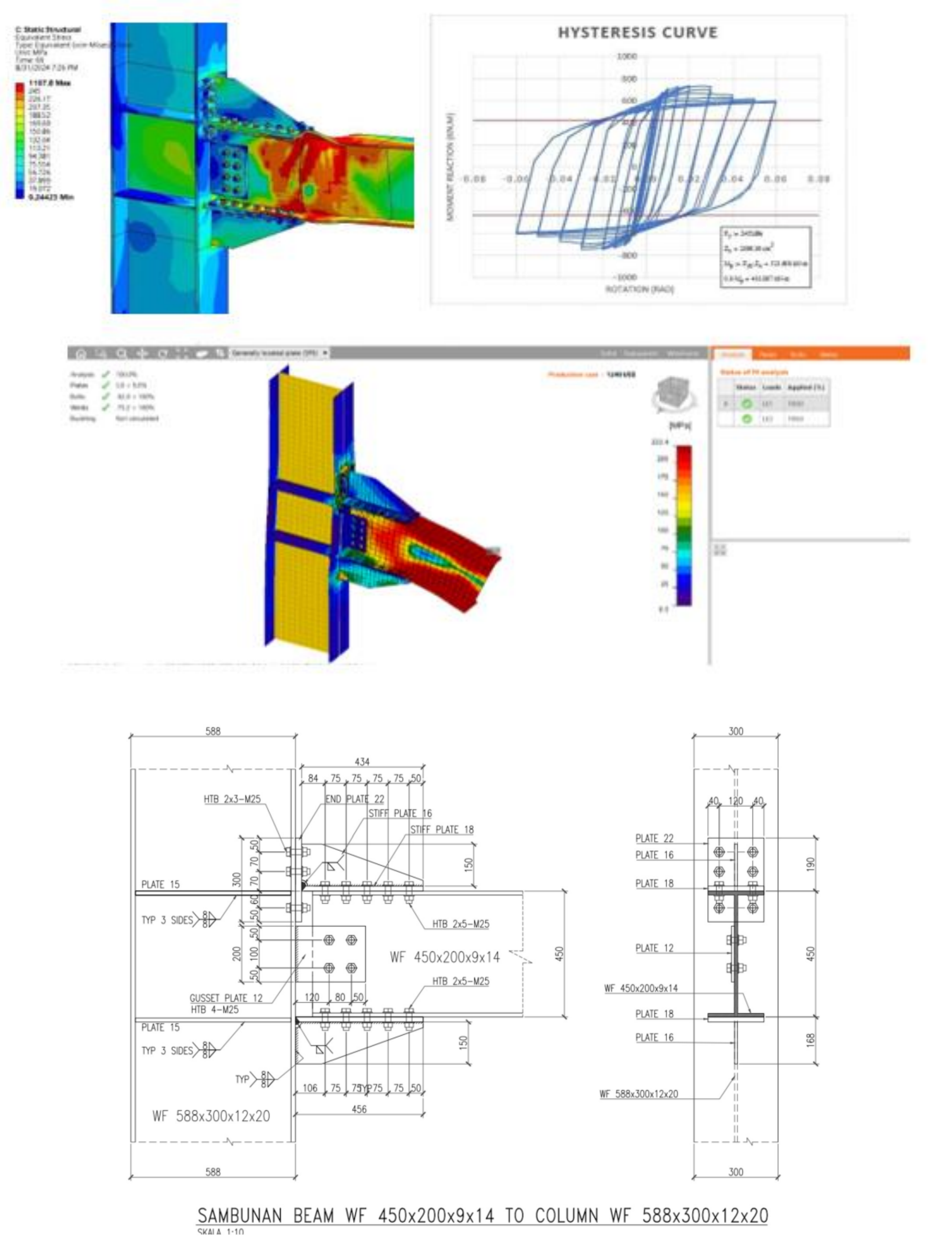

Figure 2: Behaviour hysteresis curve of model 2 Ansys and Idea Statistical

Figure 3 behavior Modified T Stub model 3 shown moment and rotation drift angle more than 0.06 radian, column size WF 588 x 300 x 12 x20, beam size WF 500 x 200 x 10 x 16

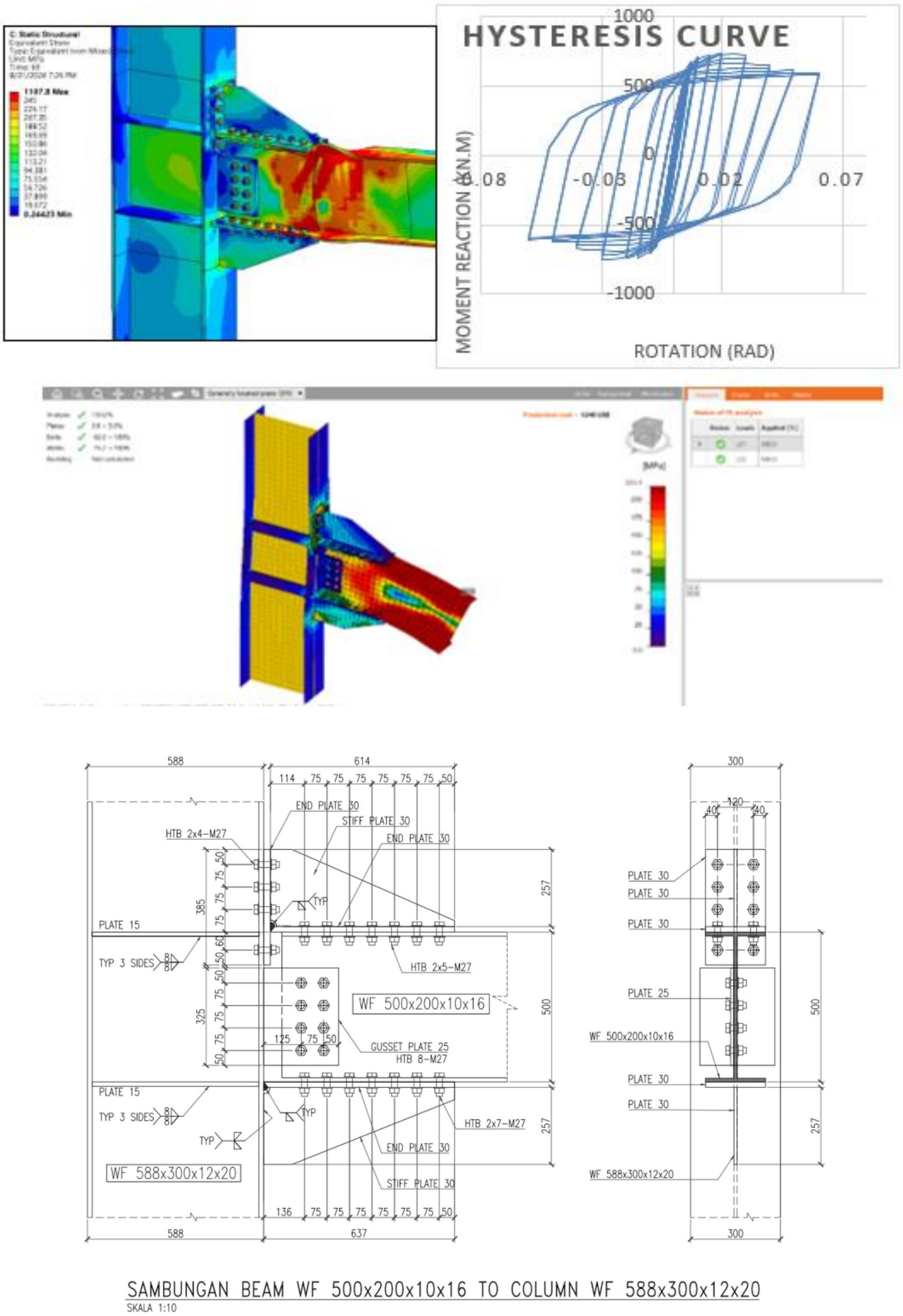

Figure 3: Behavior hysteresis curve of model 3 Ansys and Idea Statistical

Table 2: Connection performance for FE model

FE model	Moment capacity A36 (kN.m)
Model 1	104
Model 2	322
Model 3	390

Stress and Strain Contours

The plots of Von Mises equivalent stress of the modified T Stub Steel Connection

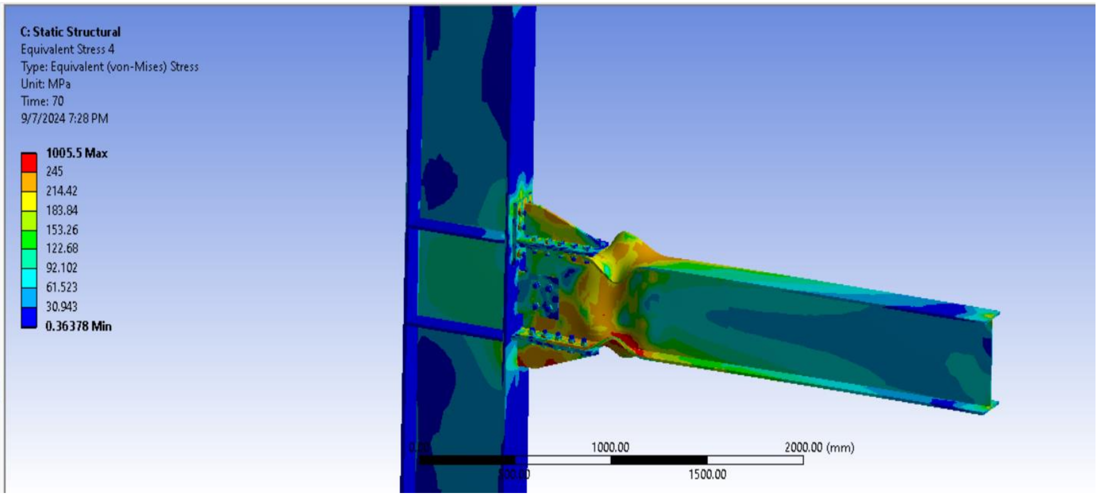

Figure 4: Stress contour of connection with A36 T Stub and beam

Model is successful in predicting maximum stress of all connection member below their ultimate stress. The maximum stress of beam is 1030 MPa and failure plastic section.

2. Conclusions

Numerical Approach Modified T Stub Steel connection following conclusions drawn.

(1) Behavior Modified T Stub Steel connection model 1, from graphic hysteresis the drift angle is 0,06 radian, so the type connection is Special moment frame prequalification connection

(2) Behavior Modified T Stub Steel connection model 2, from graphic hysteresis the drift angle is 0,06 radian, so the type connection is Special moment frame prequalification connection

(3) Behavior Modified T Stub Steel connection model 3, from graphic hysteresis the drift angle is 0,06 radian, so the type connection is Special moment frame prequalification connection.

References

[1] AISC 360. (2016). Specification for Structural Steel Buildings.

[2] Chia-Ming Uang, B., Member, S., Noel, S., & Gross, J. (2000). Cyclic Testing of Steel Moment Connections Rehabilitated with RBS or Welded Haunch.Journal of Structural Engineering, 57–68.

[3] Dewobroto, W. (2016). STRUKTUR BAJA - Perilaku, Analisis & Desain - AISC 2010 (2nd ed.). Jurusan Teknik Sipil UPH.

[4] Fema - 350. (2000). Recommended Seismic Design Criteria for New Steel Moment-Frame Buildings. www.seaoc.org

[5] Gere, J. M., & Goodno, B. J. (2012). MECHANICS OF MATERIALS (Brief).
Global Engineering.

[6] Rafezy, B., Huynh, Q., Gallart, H., & Kheirollahi, M. (2015). An Innovative Method for the Seismic Retrofitting of Existing Steel Moment Frame Structures Using Side Plate Technology.

[7] S. Taufik and R.Y. Xiao. (2006). Three-Dimensional Finite Element Modelling of Flush End Plate Connection with High Strength Steel. In G. M. and R. M.B.H.V. Topping (Ed.), Proceedings of the Eighth International Conference on Computational Structures Technology. Civil-Comp Press.

[8] Saberi, H., Kheyroddin, A., & Gerami, M. (2016). Welded haunches for seismic retrofitting of bolted T-stub connections and flexural strengthening of simple connections. Engineering Structures, 129, 31–43. https://doi.org/10.1016/j.engstruct.2016.09.050

[9] Sato, A., Newell, J. D., & Uang, C.-M. (2008). Cyclic Behavior and Seismic Design of Bolted Flange Plate Steel Moment Connections is graduate student researcher, depart-ment of structural engineering. Engineering Journal.

[10] SNI 7860:2020. (2020). Ketentuan Seismik Untuk Bangunan Gedung Baja Struktural. www.bsn.go.id

[11] SNI 7972:2020. (2020). Sambungan Terprakualifikasi Untuk Rangka Momen Khusus dan Menengah Baja Pada Aplikasi Seismik. www.bsn.go.id

[12] Sutjipto, S., & Sumeru, I. (2019). Sambungan Terprakualifikasi Menurut RSNI 7972:20xx (Adopsi AISC 358-16/358s1-18).

[13] Taufik, S., & Baharom, S. (2013). Finite element model for the behavior of partially restrained high strength web angle connections. Research Journal of Applied Sciences, Engineering and Technology, 6(6), 950–960. https://doi.org/10.19026/rjaset.6.3997

[14] Taufik, S., Baharom, S., & Xiao, R. Y. (2011). Predicted Behaviour of Partially Restrained Connection with Cold Formed High Strength Steel by 3D Finite Element Modelling. Advanced Materials Research, 250–253, 1734–1743. https://doi.org/10.4028/www.scientific.net/AMR.250-253.1734

[15] Taufik, S., & Sugianto, A. (2021). Analysing Numerical Modelling Behaviour of Reinforced Concrete Deep Beam with Strut-and-Tie Model. New Approaches in Engineering Research , 12(3), 17–34. https://doi.org/10.9734/bpi/naer/v12/3867f

[16] Taufik, S., & Xiao, R. Y. (n.d.). Numerical Modelling of Semi-Continuous Composite Connection with High Strength Steel.

[17] Tethool, Y. C. V. (2019). Perilaku Seismik Sistem Rangka Baja Pemikul Momen Khusus Menggunakan Reduced Beam Section. Musamus Journal of Civil Engineering, 2(01), 1–6. https://doi.org/10.35724/mjce.v2i01.2172

[18] Tjitradi, D., & Taufik, S. (2022). Ansys Modelling Behaviour of The Reinforced Concrete Beam with The Effect of Various Reinforcement Type and Concrete Strength. International Journal of Civil Engineering and Technology (IJCIET), 13(9), 13–22. https://iaeme.com/Home/issue/IJCI