IASMiRT

SMiRT 26
50 Anniversary Berlin/Potsdam

Transactions, **SMiRT-26**
Berlin/Potsdam, Germany, July 10-15, 2022
Division V

APPLICATION OF ANALYSIS FOR ASSEMBLY OF INTEGRATED COMPONENTS TO STEEL MEMBER CONNECTIONS FOR SEISMIC SAFETY ASSESSMENT OF PLANT STRUCTURE – PART 2: PLASTIC ANALYSIS

Akemi Nishida[1], Takahiro Murakami[2], Akira Satoda[3], Yuya Asano[3], Zhi Hong Guo[4], Masami Oshima[5], Keisuke Matsukawa[5], and Norihiro Nakajima[6]

[1]Senior Principal Researcher, Japan Atomic Energy Agency, Ibaraki, JAPAN (nishida.akemi@jaea.go.jp)
[2]Section Leader, Chiyoda Corporation, Kanagawa, JAPAN
[3]Senior Engineer, Chiyoda Corporation, Kanagawa, JAPAN
[4]Senior Manager, Surigiken Co., Ltd., Tokyo, JAPAN
[5]Senior Consultant, Chiyoda Corporation, Kanagawa, JAPAN
[6]Supervisor to Collaboration Coordinator, Japan Atomic Energy Agency, Chiba, JAPAN

ABSTRACT

From the viewpoints of safety assessments of structures such as nuclear plants, etc., seismic design of the steel structures which support critical equipment, and model analysis to predict its behavior as realistic as possible on occasion of a huge earthquake, has become more important than ever before. Especially, requirements for performance design under large earthquakes in international design codes such as ASCE 7, which are frequently adopted in recent projects, becomes more specific and quantitative. Such structural frame models are necessary to have the connection model of which non-linear behavior in the plastic region is properly described, in order to achieve reasonable amount of computation in practical design work.

The authors presented in the previous paper (Nishida et al., (2019)) that advanced FEA method called FIESTA (Finite Element Analysis for Structure of Assembly) was adopted for the steel member connections of a model plant, and that numerical experiments were conducted on the structural strength and stiffness and those results were compared with the structural frame model being used in actual design projects.

In this paper, the confirmation of the validity of the advanced FEA method are re-presented including behaviors in the plastic region by comparison with experimental results. The validated analytical method is regarded as numerical experiments to clarify the elasto-plastic behaviour of the connection, which is a strong tool to identify the inelastic deformation capacity in the classification of moment frame systems in American seismic design codes such as ASCE 7 and AISC 341.

INTRODUCTION

The safety under a huge earthquake event for any plants dealing chemical process, ranging from the most critical one, nuclear plants, to popular ones, various facilities in oil and gas industries, have been the highly concerned matter publically, not only of the authors' affiliations. For steel structures supporting critical equipment or basic lifelines, seismic design methodology reflecting advanced computational technologies is being developed and applied for building new plants worldwide. In addition, maintaining the existing "aging" plants has become important in countries like Japan and those advanced computational technologies are expected to be applied in seismic assessment of structures in order to see impacts more accurately, and quantitatively.

Seismic design method has been adopting these advanced calculation in international design codes, for example, ASCE 7-16, “Minimum Design Loads and Associated Criteria for Buildings and Other Structures”, increased the number of input earthquake motions from 7 to 11 in an optional method of non-linear time history analysis. At the same time, the performance design philosophy is reflected into measurable and quantitative criteria in many codes and standards. The structure’s behavior beyond the steel yielding point at a large earthquake relies heavily on the response behaviour of the connections and analysis of the structural frame models having the connection of nonlinear functions is considered to be the key.

Japan Atomic Energy Agency (JAEA) had strong interests on these issues, and developed FIESTA (Finite Element Analysis for Structure of Assembly) to discipline the connection assembly analysis. The authors had reported a steel structure of a petrochemical plant built into a finite element model as an assembly of integrated components. Time-history response analysis was conducted with high-performance computer technology to show the feasibility of this analysis method (Nakajima et al. (2015)). The essence of this highly-sophisticated finite element analysis (FEA) was adopted for the steel member connections of a structure. Nishida (2006) also discussed a matter of fundamental function of FIESTA.

The authors take the same connection model as those used in the last SMiRT 25 paper (Nishida et al., (2019)), by using the advanced three-dimensional modelling method of FIESTA. While the last paper discussed within the elastic range, the analysis extends to plastic range to perform non-linear analyses to reveal the mechanism of the bolted connection assembly. The validity of the analytical method is confirmed by comparison with experimental results. The validated analytical method is regarded as numerical experiments to clarify the elasto-plastic behaviour of the connection, which is a strong tool to identify the inelastic deformation capacity in the classification of the moment frame systems in American seismic design codes such as ASCE 7 and AISC 341, “Seismic Provisions for Structural Steel Buildings”.

MODELS FOR NUMERICAL EXPERIMENT

With regard to the verification of the validity of the analysis method extended toward the plastic range, the experimental study paper by Sumner (2000) conducted in the SAC steel project was referred again. The cyclic bending tests of bolted moment endplate connections of the same type (Specimen ID: 4E-1.25-1.5-24, endplate thickness 38 mm (1.5 in)) that we adopted in the previous paper, was taken for study. Additionally the test on another specimen of thinner endplate (Specimen ID: 4E-1.25-1.125-24, endplate thickness 29 mm (1.125 in)) was used for verification. Those two test specimens are called as type A and type B, respectively, and the configuration of those test specimens is common and shown in Figure 1.

In the real experiment, forced displacement was applied cyclically and the displacement was increased gradually for the beam-column connection to advance from elastic to plastic region. The forced displacement was applied at the tip of the beam and the resultant behaviour at the beam-column connection is presented in terms of moment vs total rotation diagram, for example of type A specimen, as Figure 2.

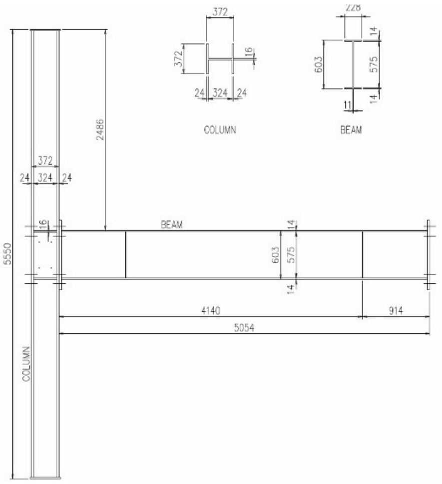
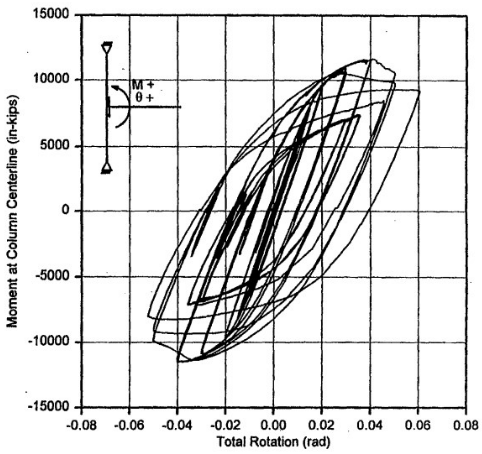


Figure 1 A draft of a fundamental model for numerical experiment



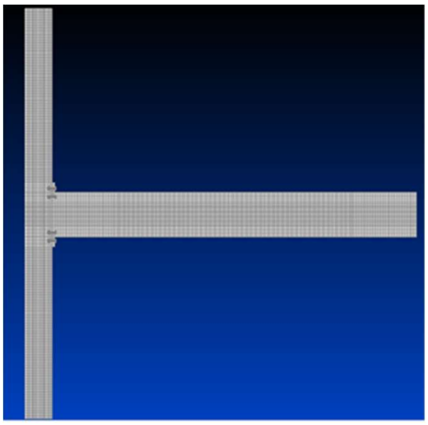
(Ref.: Sumner (2000))

Figure 2 Moment vs total rotation

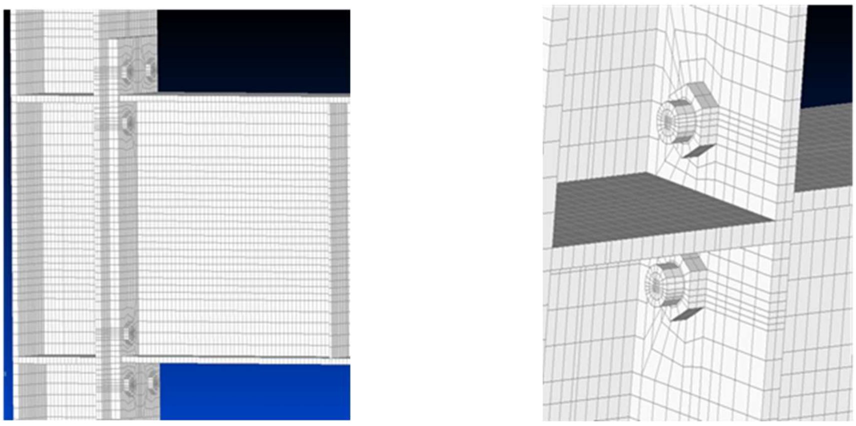
NUMERICAL EXPERIMENTS

Finite Element Model

The finite element model is identical to that built in the previous paper, shown in Figure 3. The bolted connection model is adopted and calculated by ABAQUS. Characteristics of the advanced FEA methodology are incorporated; that separate components such as bolts, nuts, and plates, have boundary of contact element surfaces attached by bolt's tensile force, whereas the welded parts such as the beam and the endplate are built as continuous elements. The top and bottom of the column are completely restrained.



(a) Overview of a model



(b) Connection parts of a model (c) Bolts tied parts of a model

Figure 3. A finite element meshed model for numerical experiment

Parameter fitting

In order to simulate the test result, three sinusoidal curve cycles of forced displacement of which amplitudes are equivalent to story drift of approximately 0.02, 0.04 and 0.06 radian, were applied at the tip of the beam.

In the elastic analysis, steel material properties of generic structural steel, SS400 are assigned. Young's modulus is set to 210,000 MPa, Poisson's ratio is to 0.3, and density is to 7.8×10^{-9} ton/mm^3. In the elasto-plastic analysis, material properties in the plastic region are described to follow a multi-linear model shown in Figure 4, where F_y: yield stress, F_u: ultimate stress, E: elastic stiffness, E_t: plastic stiffness, are calculated. The plastic stiffness beyond the yield point is reduced to E_t being one tenth of the elastic modulus, and after passing the ultimate strength F_u, the softening occurs at the rate of $-E_t/2$. The curve representing this plastic softening beyond the ultimate strength was adopted because the results simulate the experiments better in some points such as maximum moment at the connection part, plastic rotation angle, and bolt tension especially some being retained after unloading process. Material elsto-plastic properties are dependent on steel material specifications and summarized as shown in Table 1.

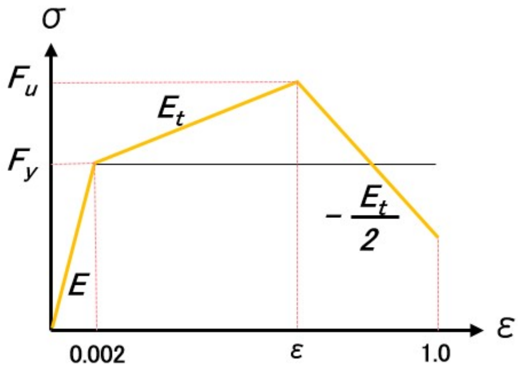


Figure 4 Steel material property curve in the plastic region

Table 1 Steel material parameters for components

Component	Material spec.	F_y (MPa)	F_u (MPa)
Beam/Column	ASTM A572 Gr. 50	370	487
Bolt, Type A	ASTM A490	890	1193
Bolt, Type B	ASTM A325	558	724

Results – type A

The simulation of the type A connection is presented in terms of moment vs total rotation in Figure 5. The curve for the experiment is also drawn by tracing the centreline or so-called "backbone" curve in the elastic region and the envelop in the plastic region.

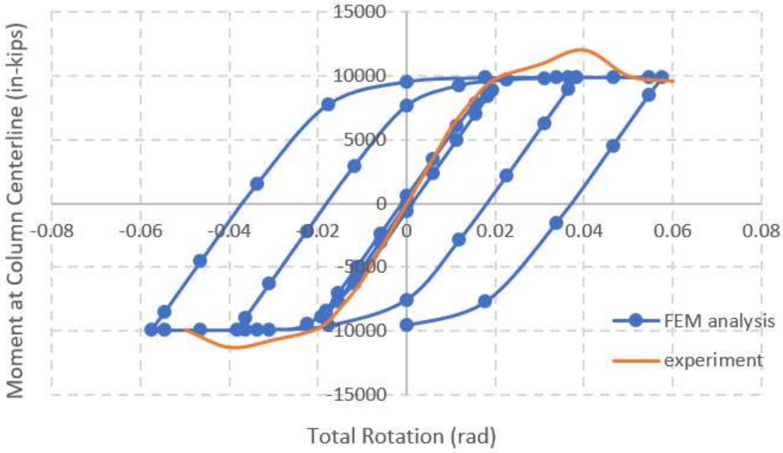


Figure 5. Comparison of experiment with simulation, Moment vs Total rotation

The bolt axial tension in the test is simulated as shown in Figure 6, and test data is also shown. Initial bolt tension seems to remain at reduced amounts in the course of unloading-reloading paths. Regarding comparison of the FEA results with the test results, good correlation is obtained for bolts.

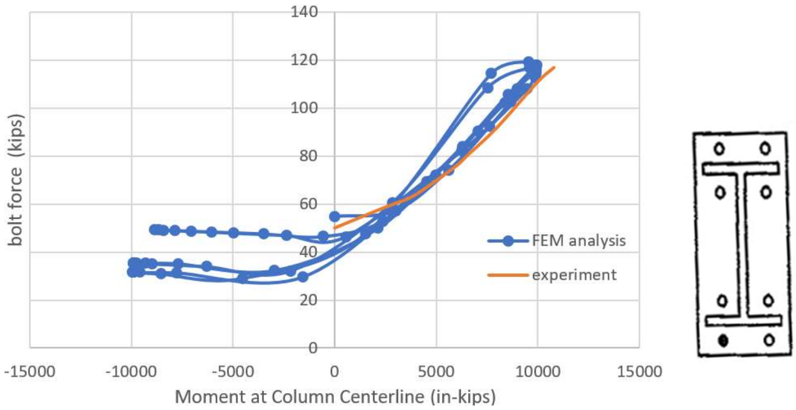


Figure 6. Comparison of experiment with simulation, bolt axial tension

Graphical outputs of FEM simulation, i.e., Figure 7, and Figure 8 shows the maximum Von Mises stress and the maximum plastic energy dissipation for the test, respectively.

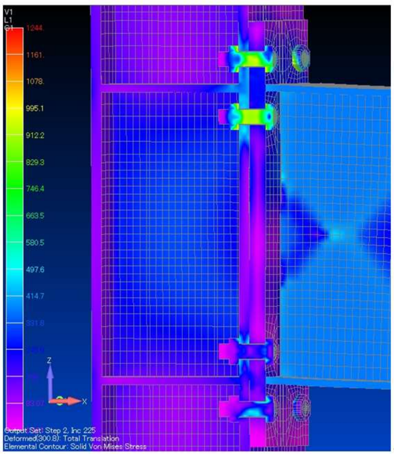


Figure 7 Maximum von Mises stress

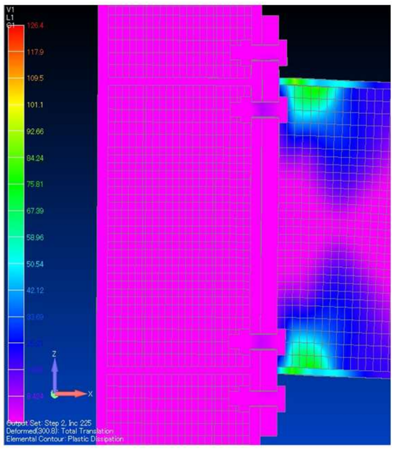
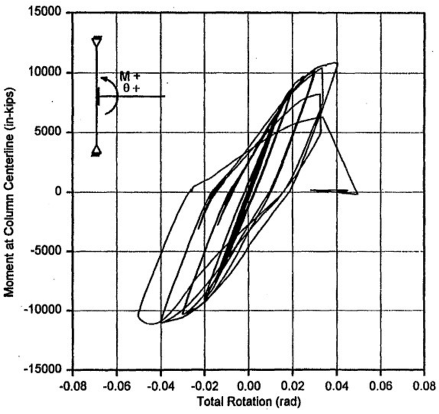


Figure 8 Maximum plastic energy dissipation

According to the SAC report, the type A test resulted in a failure mode of local flange and web buckling of the beam starting at total rotation of 0.04 radian and no distress was observed within the connection region. From Figure 7, Von Mises stress is observed high at the upper bolt, however its utilization ratio was not so high for Gr. 50 bolt of high tensile strength. It is rather noticeable that stress accumulated in the beam, and Figure 8 shows significantly high value of plastic energy dissipation in the beam parts near the connection point, while does not show any high value in the endplate and column. It is a good prediction that failure occurs in the beam, not in the connection.

Results – type B

Figure 9 shows the test result of type B specimen, and Figure 10 shows the FEA results, both in terms of moment vs total rotation. Three curves of the test, i.e., the "backbone" curve, outmost loading-unloading path curve, and 2nd outer path curve, are superimposed in Figure 10 for comparison.



(Ref.: Sumner (2000))

Figure 9 Moment vs. total rotation of the experiment

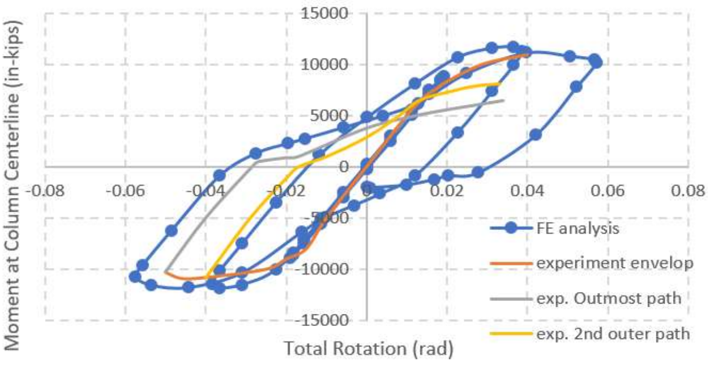
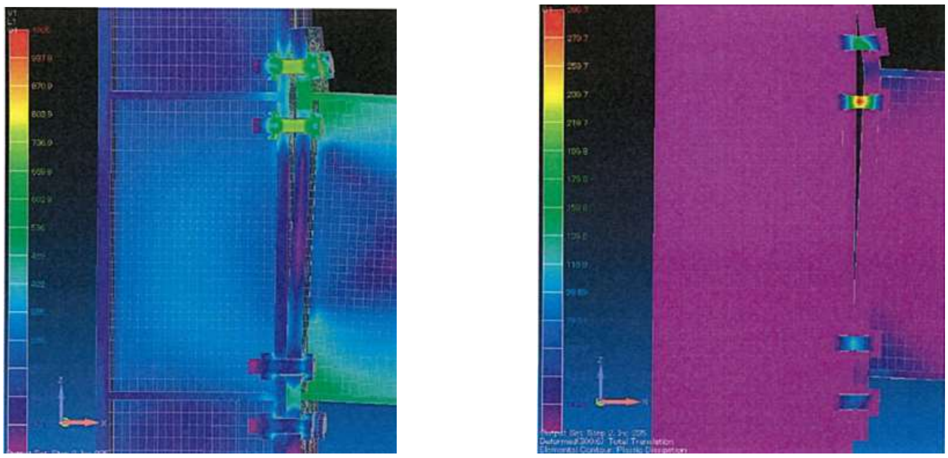


Figure 10 Moment vs. total rotation of the FEA

Figure 11 shows FEA results of Type B specimen in terms of (a) Von Mises stress and (b) plastic energy dissipation, both at total rotation of 0.06.



(a) Von Mises stress (b) plastic energy dissipation

Figure 11 Type B specimen, FEA results at total rotation of 0.06

According to the SAC report, Type B specimen resulted in a failure mode of endplate yielding followed by bolt tension rupture. The bolts in the type B test were observed to behave similar to those of type A during the lower loading steps, however to exhibit a sharp increase in bolt strain which corresponds with the initial yielding of the endplate. The reason of the failure mode difference is mostly attributed to the bolt material, that type B bolt is ASTM A325 having F_y of 558 MPa, which is smaller than type A of ASTM A490, F_y of 890 MPa.

From Figure 11, it is apparent that significant plastic deformation occurred in the bolts and the endplate nearby, and a gap was observed between the column and the endplate. High plastic energy dissipation is indicated in the upper bolts while almost no color change in the beam side. In Figure 10, a drop of strength and an increase of rotation are observed in the last two loading-unloading path in the test, due to the plastic deformation of the bolt and endplate mentioned above. These behaviors are well simulated in the advanced FEA as shown in Figure 11.

Verification

Type A and type B specimens in SAC project were planned to have the ratio of the design plastic moment of the connection to that of the beam being set to 1.0 and 0.95, respectively. It means that type A have the connection of as almost the same plastic capacity as the beam, and type B have a slightly weaker connection than the beam. The advanced FEA methodology differentiates the structural behaviours of two connections in the plastic region, apparently by graphical presentation of Von Mises stress and plastic energy dissipation. As quantitative presentation of the test in terms of moment vs total rotation relationship, bolt tension simulates the experimental data, the parameters of material properties especially in the plastic region are considered reasonably determined. Based on the above, the advanced FEA methodology adopted in the study is deemed verified to sufficiently simulate the structural performance of the steel connection.

POTENTIAL APPLICATION TO STRUCTURAL DESIGN

Connection performance in structural frame analysis

Although the advanced FEA utilizing the finely built mesh model is verified to be accurate, it is not feasible to build a model of whole structure and is necessary to decrease the model size with sufficient accuracy in order to be utilized in practical design work. The elasto-plastic performance of the beam-column connection confirmed by the advanced FEA can be described into a non-linear element. For example, the test specimen of the connections studied in the previous sections is modelled as shown in Figure 12.

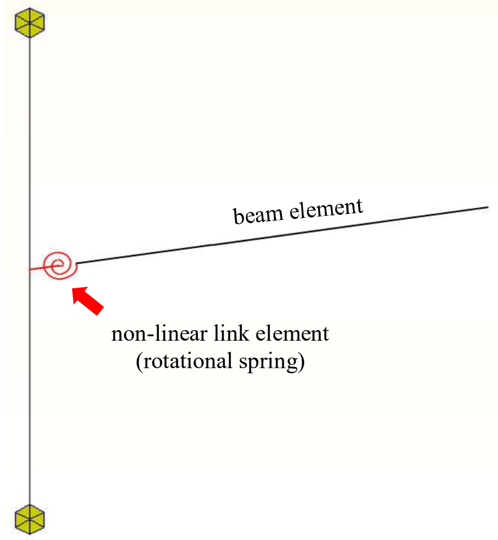


Figure 12 structural frame model of the experiment

In a model shown in Figure 12, the beam and the column are expressed by beam elements and a generic link element is set to connect them. As the experimental program focused on the order of yielding and on their plastic behavior separately, a non-linear link element as the connection is necessary. The beam element can rather easily described with simple non-linear function such as a bilinear curve automatically calculated by its section shape and dimensions. The generic link element shall simulate the elasto-plastic behaviour confirmed by the advanced FEA thus the non-linear function becomes significantly complex because the connection consists of parts assembled, which is an issue for building a frame analysis model.

Potential application for structural design

The seismic design of steel frame structure in accordance with American codes can be summarized as follows. The special seismic design is necessitated when a target structure is to be built in earthquake-prone regions stated in ASCE 7-16. Then the Seismic Force Resistant System (SFRS) regulated by the seismic design code of AISC 341 is used. SFRS is further classified into three systems, Ordinary Moment Frame (OMF), Intermediate Moment Frame (IMF), and Special Moment Frame (SMF), in the order of levels of its inelastic deformation capacity. That is, the beam to column connection is required to sustain higher story drift angle as the system goes from OMF to IMF, further to SMF. To design higher inelastic rotation, not only Fully Restrained (FR) connection but Partially Restrained (PR) connection are permitted in all three MF systems. There are many requirements of performance parameters in the plastic region for those connections and for IMF and SMF, those have to be designed in accordance with AISC 358, “Prequalified Connections for Special and Intermediate Steel Moment Frames for Seismic Applications”. In some cases, the connection performance is required to be demonstrated by analysis. Determination or confirmation of the non-linear parameters of the connections requires highly detailed analysis.

Requirements of the beam-to-column connection for each moment frame, OMF, IMF, and SMR, are summarized in Table 2.

Table 2 Requirements for connections of different SFRS

	Capacity of story drift angle	Flexural resistance	Method of demonstration
OMF-PR	N.A.	Not less than 0.5 $M_{p\ beam}$	N.A.
OMF-FR	N.A.	Not less than 1.1 $M_{p\ beam}$	N.A.
IMF	At least 0.02 rad	At least 0.8 $M_{p\ beam}$@0.02 rad	ANSI/AISC 358 Prequalified design Cyclic bending test
SMF	At least 0.04 rad	At least 0.8 $M_{p\ beam}$@0.04 rad	

Note : $M_{p\ beam}$ is the moment to cause full plastic state of the connecting beam.

In AISC 341, OMF are expected to provide only minimal levels of inelastic deformation capacity. Thus the general intent of the OMF design provisions is that non-ductile connection failure should not be the first significant inelastic event. In contrast, SMF is generally expected to experience significant inelastic deformations during large seismic events. As it goes down in Table 2, the performance of the connection becomes more ductile and its design and detailing requirements becomes more restrictive. AISC 358 describes ten different connections that have been prequalified for use in both IMF and SMF systems. The Bolted Unstiffened Extended End Plate (BUEEP) is one of those prequalified design to which the type A and B belong. Even with the prequalified design, AISC 341 clearly stated that partially restrained (PR) connection is permitted in SMF system but is required to be demonstrated by analysis.

The discussion above reveals that design of steel structure against large earthquakes advances from conventional connections of binary boundary conditions of either FIX or PIN, to connection's performance parameters ranging to the inelastic region quantitatively. Essential parameters such as the rotational stiffness in the elastic region, the maximum resistance, and the plastic rotation capacity, are macroscopic parameters of the connection, which is composed of multiple parts of bolts, endplates, and stiffeners. In order to define those parameters relative to the capacity of the beam to be connected, the advanced FEA method is considered effective means.

There are drawbacks and issues to be confirmed by research with regard to the connection performance. The cyclic bending is only on the direction of the beam's strong axis and issues remain on connection's responses to multidirectional force even though the whole frame is examined from other directions. Bolted connection is more prone to construction allowance than welded connection and its impacts on the model analysis has to be evaluated. However the authors believes that a more precise and quantitative description of the performance of the connection detail is the way forward to practical design work assisted by high performance computation technology.

In addition, the approach would work for evaluation of existing structures against large earthquake which is especially critical in Japan. The advanced FEA can be applied to confirm the connection performance by setting specific safety factors for aging such as deformation due to modifications, and loading history. Ordinary frame analysis assisted by the advanced FEA will provide accurate evaluation of the operating plant and basic information for risk assessment.

CONCLUSION

In this paper, the study on the advanced FEA i.e., FIESTA, of the connection model tracing the bending test of the steel column-beam connection of actual scale in the SAC project is extended to the plastic region and the following conclusions are drawn.

The advanced FEA methodology is applied to simulate the bending test of the endplate type beam – column bolted connections. Steel material inelastic parameters are able to calculate the behaviour of the test in the plastic region appropriately. By evaluating the simulation result macroscopically in terms of rotational stiffness and its softening, and microscopically in terms of stress and energy dissipation, by comparison with the experiments, the methodology is verified to describe the actual phenomena properly.

As the proposed approach of utilizing the advanced FEA could provide connection's parameters sufficiently accurate for the frame analysis, it can be regarded as numerical experiment. The results of the numerical experiment can be reflected into the parameter setting of the inelastic rotational spring element between column and beam and incorporated frame model can be analysed to simulate the inelastic behavior of whole steel structure.

In American steel design code system, earthquake-resistant structure system such as Special Moment Frame (SMF) is authorized and the key element is the inelastic performance of the connection that is required to perform more ductile, i.e., to sustain higher drift angle. As well as AISC 358 prequalified design method, testing, and quality of welding, the advanced FEA is considered effective means.

Although there yet remains some issues, the authors would like to promote studies and applications of the advanced analytical technique to the practical engineering to both earthquake resistant design of new structures and evaluating expected performance of the existing structure at a large earthquake.

ACKNOWLEDGMENTS

This research was conducted with the supercomputer ICE X in the Japan Atomic Energy Agency.

REFERENCES

ANSI/AISC 341-16, (2016), "Seismic Provisions for Structural Steel Buildings", AISC

ANSI/AISC 358-16, (2016), "Prequalified Connections for Special and Intermediate Steel Moment Frames for Seismic Applications", AISC

ASCE/SEI 7-16, (2016), "Minimum Design Loads and Associated Criteria for Buildings and Other Structures", ASCE

Fish, J., and Belytschko, T., (2008), "A First Course in Finite Elements (ABAQUS Student Edition)," Maruzen (Translated to Japanese edition by Yamada, K)

Mays, T. W., (2000). "Application of the Finite Element Method to the Seismic Design and Analysis of Large Moment End-Plate connections," Dissertation to Virginia Polytechnic Institute & State University.

Nakajima, N. et al., (2013), "Assembly Structural Analysis System," Paper No. 814, Transactions, SMiRT-22, San Francisco, California, USA - August 18-23, Division III

Nakajima, N. et al., (2015), "Time Domain Response Analysis for Assembly by Integrating Components," Paper No. 377, Transactions, SMiRT-23, Manchester, UK - August 10-14, Division III

Nishida, A. et al., (2019), "Application of Analysis for Assembly of Integrated Components to Steel Member Connections towards Seismic Safety Assessment of Plant Structures," SMiRT-25, Charlotte, NC, USA, August 4-9, 2019

Nishida, A. et al., (2006), "Numerical Simulation System Three-Dimensional Virtual Vibration Testbed," Trans. Atomic Energy Society Japan, 6(3), 376-382

SAC Joint Venture. (2000). "State of the Art Report on Connection Performance," FEMA-355D, Federal Emergency Management Agency.

Sumner, A. E., et al., (2000). "Cyclic Testing of Bolted Moment End-Plate Connections," SAC Joint Venture, Report No. SAC/BD-00/21.