Quasi-Static Analysis of End Plate Beam-to-Column Connections

A. Al-Rifaie, Z. W. Guan, S. W. Jones

***Abstract*—**This paper presents a method for modelling and analysing end plate beam-to-column connections to obtain the quasi-static behaviour using non-linear dynamic explicit integration. In addition to its importance to study the static behaviour of a structural member, quasi-static behaviour is largely needed to be compared with the dynamic behaviour of such members in order to investigate the dynamic effect by proposing dynamic increase factors (DIFs). The beam-to-column bolted connections contain various contact surfaces at which the implicit procedure may have difficulties converging, resulting in a large number of iterations. Contrary, explicit procedure could deal effectively with complex contacts without converging problems. Hence, finite element modelling using ABAQUS/explicit is used in this study to address the dynamic effect may be produced using explicit procedure. Also, the effect of loading rate and mass scaling are discussed to investigate their effect on the time of analysis. The results show that the explicit procedure is valuable to model the end plate beam-to-column connections in terms of failure mode, load-displacement relationships. Also, it is concluded that loading rate and mass scaling should be carefully selected to avoid the dynamic effect in the solution.

***Keywords*—**Quasi-static, end plate, finite element, connections.

I. Introduction

End plate connections including flush plate and partially depth plate are the most popular steel bolted connection types in use for structural building frames in the UK because they could be fabricated easily with low cost [1]. The work presented here is a part of a work to study the lateral impact response of end plate connections. However, the quasi-static behaviour is needed in this study to be compared with impact response. Besides, the experimental tests are always time consuming and expensive, which can be reduced for validation purposes only, and the validated model could be used then for investigating different parameters.

Non-linear static analysis was used by many researchers to model the bolted connections under quasi-static loading. Due to the limitations of computational capabilities, the first attempts were developed using two-dimensional (2D) models [2]-[4]. It was found that the 2D models are unable to predict accurately the connection behaviour. Later and with the development of computers, many studies were carried out using three dimensional-implicit procedure (3D) models considering various geometries [5]-[7]. However, this concept may result in convergence difficulties at which the model is aborted. This is due to the complex geometry and contact surfaces that such connections contain. The state-of-the-art-finite elements modelling is adopted using dynamic analysis to model the quasi-static behaviour to avoid convergence difficulties and geometric complexities. Abdullah et al. [8] presented a method for modelling steel deck-concrete slabs under quasi-static load using non-linear dynamic finite element analysis. The numerical results showed good correlation with the experimental results. Natario et al. [9] also used the explicit integration to evaluate the web crippling behaviour of cold-formed steel beams under quasi-static load. It was concluded that this type of analysis is effective to model web crippling failure in terms of load-displacement curves and failure modes.

In this study, the dynamic analysis using ABAQUS/Explicit is employed to model the end plate connections under quasi-static loading. This concept was chosen because it is the appropriate method for models having many contact surfaces [10]. Also, it is a good opportunity to investigate the validity of this concept to model bolted connections after confirming its validity for other structural members as mentioned in references [8], [9]. A three dimensional non-linear finite element model of end plate connection was developed. Numerical results were verified then with the experimental test results in terms of load-displacement relationships, strain-displacement relationships and failure modes. Factors affecting the model accuracy were also discussed.

II. Description of the Finite Elements (FE) Model

A three dimensional (3D) FE model was generated for the two types of end plate connection using ABAQUS software to investigate the quasi-static behaviour of such connections loaded laterally on the column. This model is built to compare the behaviour of connections under quasi-static with that under impact load and to study the dynamic effect on their components. Fig. 1 shows the entire model and its individual components. The size of the beam and column were 305x127x37 and 152x152x37, respectively. An 8 mm thick end plate was welded to the bottom of the beam then connected by four M16 high strength bolts. All parts were modelled using the real dimensions, except bolts and nuts. The modelling of the bolt with threads is avoided by creating an equivalent hole of 40 mm diameter along the bolt. The hexagonal nuts were modelled using an equivalent circle. Half model is used due to symmetry to save the time of analysis.

A. Solution Strategy

Two different solution concepts are available in ABAQUS: the implicit analysis and explicit procedure. The implicit analysis depends on static equilibrium, using global stiffness

A. Al-Rifaie, Z. W. Guan, and S. W. Jones are with the Civil Engineering and Industrial Design Department, University of Liverpool, L69 3GQ UK (e-mail: a.k.i.al-rifaie@liv.ac.uk, zguan@liv.ac.uk, swjones@liv.ac.uk).

matrix and simultaneous solution of a set of equations. Iterations are repeated in each time increment until convergence is achieved. However, with zero tangent stiffness, this method is unable to achieve the convergence. This problem could be avoided using Riks method by allowing the load and displacement to vary throughout the time step [10]. Nevertheless, these two algorithms are unlikely to produce an effective solution if the model has complicated contact interactions, as in bolted connections.

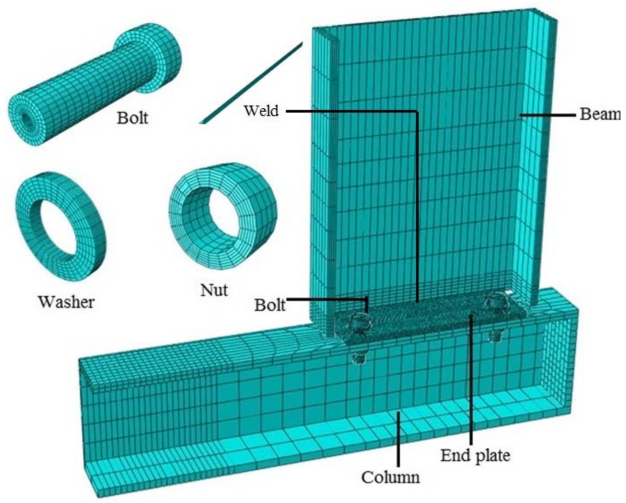


Fig. 1 FE model for the end plate connection

The explicit procedure is originally developed to simulate high-speed impact events in which inertia effect should be considered, and the convergence problems are not existing. Also, explicit procedure deals better than the implicit procedure with problems having complicated contacts. Hence, in order to simulate quasi-static behaviour using dynamic explicit procedure, the inertia effect should be controlled to be insignificant. This can be achieved by ensuring that the ratio of kinetic to internal energy does not exceed 5% for the whole model and by using appropriate loading rate and/or mass scaling factor [10]. Hence, the effect of different loading rates and mass scaling factor should be investigated to obtain the optimum values that can be used for validation of the numerical results with the experimental results.

B. Element Type

The first order-reduced integration solid element C3D8R is used in the FE model. This element allows for geometrical and material nonlinearity and it minimizes the computational expense of element calculations. Also, it is not susceptible to shear-locking phenomenon which makes the model stiffer. However, the only drawback of this element type is their sensitivity to hourglass modes at which zero stress and strain generated in the integrated point of the stressed element. This can lead to an inaccurate solution and should be addressed properly. To avoid that, four elements through the thickness of connection components that exposed large deformation (plate and bolt) are used. Also, contact constraints used in the model are distributed among several nodes using surface to surface contact algorithm.

C. Modelling of Contact

Modelling of contact between various surfaces is one of the most critical processes. Hence, considerable attention was made to select the proper master and slave surfaces and to assign appropriate interaction properties to model the interaction. Tie constrains and surface to surface contact formulations were used to model contact surfaces. The former was employed to connect weld to the beam and plate, nuts to bottom washers, and nuts to bolts, as no visible deformation was observed on these parts during the experimental tests. Penalty friction formulation with a coefficient of friction of 0.2 between contact surfaces was selected to simulate the tangential behaviour of the contact, whilst the normal hard contact allowing separation was assumed for all interaction surfaces.

D.Material Properties for the FE Model

1) Elasto-Plastic Behaviour

The Elasto-plastic behaviour of steel materials was modelled using ABAQUS/Explicit based on the engineering stress-strain curves obtained experimentally as shown in Fig. 2. Hence, Modulus of elasticity (E) and yield stress (σ_y) were taken from the curves for each material to be used as input data to define the elastic stage up to the yielding point. In order to obtain reasonable material properties in the plastic stage, the engineering stress-strain curves were modified to obtain the true stress-strain curves using the following equations:

$$\sigma_{true}=\sigma_{eng}\,(1+\varepsilon_{eng}) \tag{1}$$

$$\varepsilon_{true}=\ln\,(1+\varepsilon_{eng}) \tag{2}$$

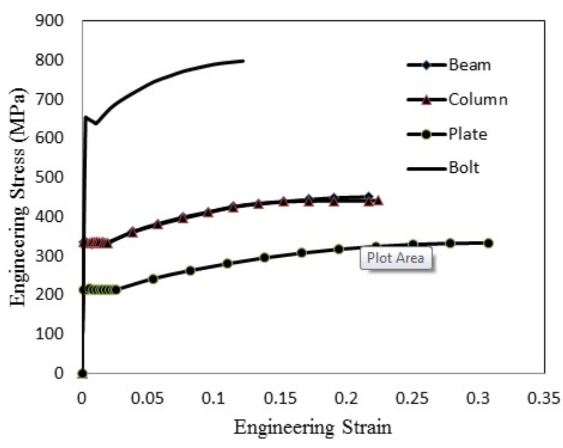


Fig. 2 Material properties of steel

2) Ductile and Shear Damage

After the strain hardening stage, onset of damage and damage evolution need to be modelled. Fig. 3 shows a typical stress strain curve with progressive damage degradation of an isotropic material. The solid line after onset of damage (Damage parameter (D) =0) represents the initiation of damage, whilst the dashed curve refers to the material response without damage. Hence, a finite element model without damage properties is unable to take into account the stiffness degradation of the element leading to over-estimated internal stresses. Consequently, damage properties were added to the connection components (bolts and plate) due to large deformation observed while the other parts were modelled using elasto-plastic behaviour only to save the time during analysis. The modelling of ductile and shear failure under quasi-static was performed using the ABAQUS keyword option. Hence, the ultimate tensile true stress of the plate and bolt obtained from the coupon tests were used as input data to model the ductile damage. Also, shear damage of the end plate was modelled in the same way as ductile damage and a value of fracture strain of 0.2 was complied with the experimental results and failure modes for all models.

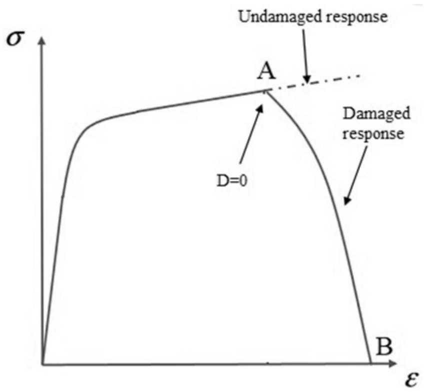


Fig. 3 Stress-strain curve with progressive damage degradation

E. Loading Rate and Mass Scaling

The load application is made using a reference point created at a rigid cubic body of 100 mm x 100 mm x100 mm. One face of the rigid body was in contact with the free end of the column to apply the load. The load is applied as a displacement control using smooth step amplitude. This option is particularly recommended due to its efficiency to reduce noise generated from the initial kinetic energy [10]. Loading rate can be defined as the ratio of the maximum displacement at the free end of the loaded column to the assumed step time. Hence, using the natural loading rate used in the experiment is ineffective because it needs a long time to finish the analysis. Thus, it is beneficial to select an appropriate loading rate at which the dynamic effect is eliminated and the analysis time is reduced. This can be achieved by adopting low loading rate (i.e. higher step time) corresponding to an appropriate mass scaling factor. More discussion is presented in Section III, to investigate the effect of these two factors and to select the optimum values that can be used in the numerical model.

III. Verification of Numerical Results with Experimental Tests

Fig. 4 shows the procedure to validate the FE model proposed in this study. The procedure begins with selecting a random step time to specify the loading rate used in the analysis by dividing the maximum displacement to the step time. This loading rate should be tested then with different mass scaling factors ensuring that the ratio of kinetic energy to internal energy of the whole model (Ek/Ei) does not exceed 5%, as mentioned in Section II.A. With this condition, the dynamic effect produced from the explicit procedure can be eliminated and the CPU time is reduced. Hence, the first trials were carried out using a mass scaling factor of 10^8 with three loading rates 10 mm/s, 2 mm/s and 0.5 mm/s. It is concluded that using high loading rate leads to increase the energy ratio corresponding to a highly localized deformation pattern in the loaded column, which is not the case in experimental deformed shape, as shown in Fig. 5 (a); while, this local deformation significantly reduced using lower loading rate, as shown in Fig. 5 (b). In spite of the matching between the experimental and numerical deformed shape using lower loading rate, the dynamic effect seemed to be unacceptable because the energy ratio (E_k/E_i) exceeds the limit aforementioned, as shown in Fig. 6 (a). This may be attributed to the use of high mass scaling factor. Therefore, three additional mass scaling factors less than 108 were investigated with a loading rate of 0.5 m/s, as shown in Figs. 6 (b)-(d). In these figures, it can be seen that the dynamic effect is reduced with lower mass scaling factor. Consequently, an initial loading rate of 0.5 mm/s and a mass scaling factor of less than 10^6 could be used to validate the experimental results considering the use of applicable CPU time.

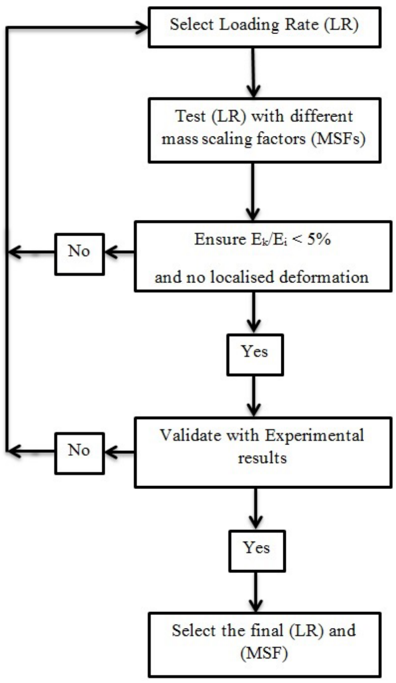
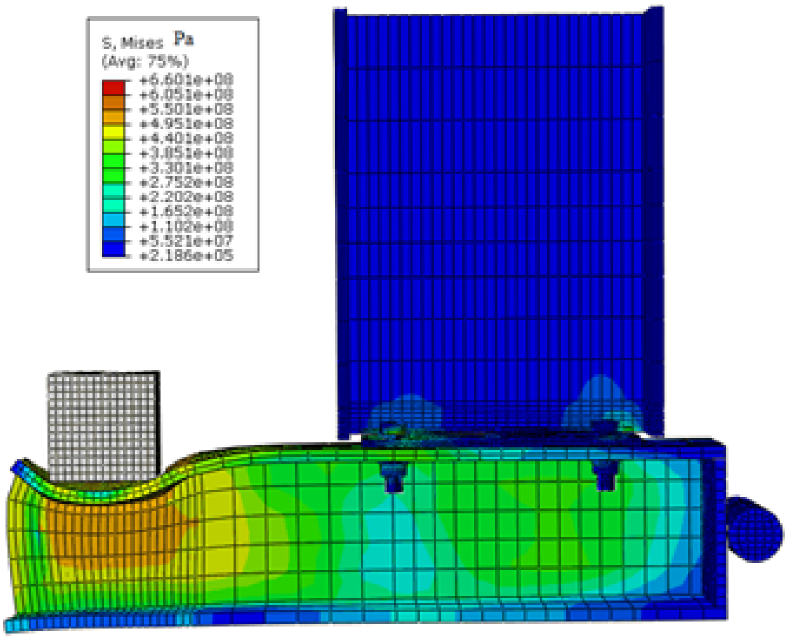
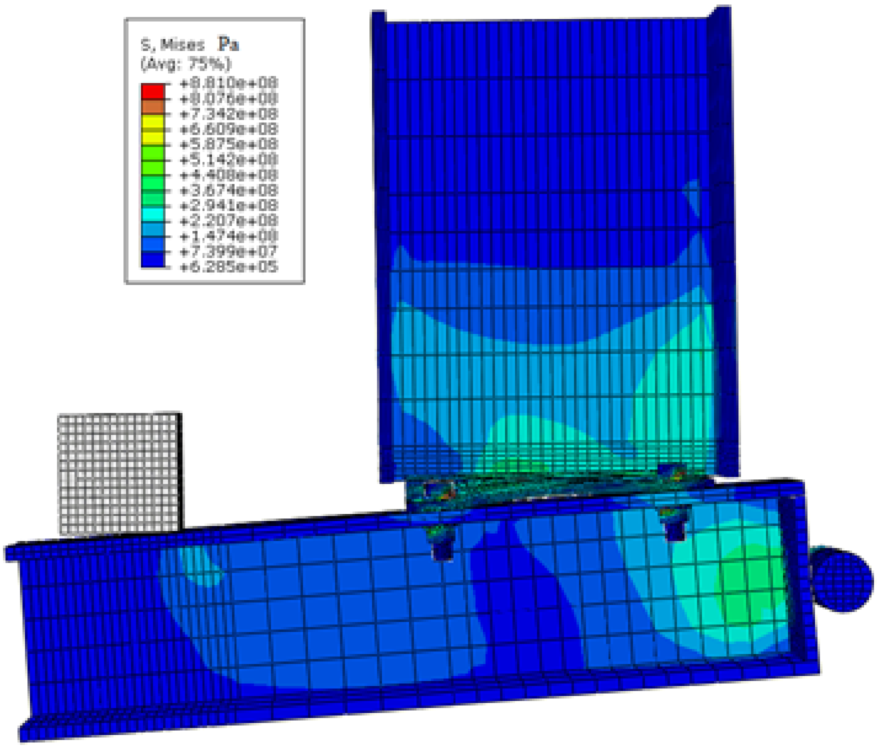


Fig. 4 Procedure of validation the numerical model



(a) Loading rate = 10 mm/s



(b) Loading rate = 0.5 mm/s

Fig. 5 Column deformed configuration with different loading rates

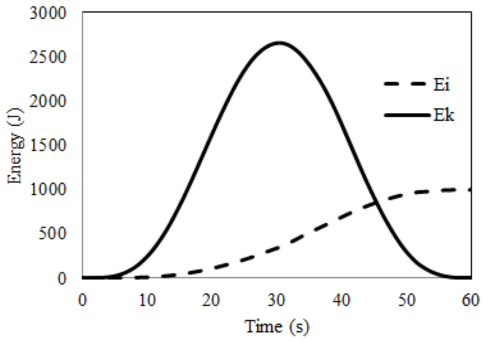


Fig. 6 (a) Energy-time relationship with LR=0.5 and MSF=10^8

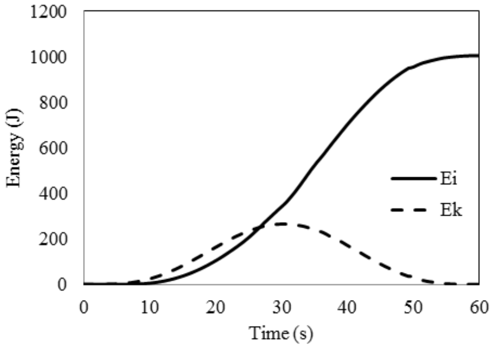


Fig. 6 (b) Energy-time relationship with LR=0.5 and MSF=10^7

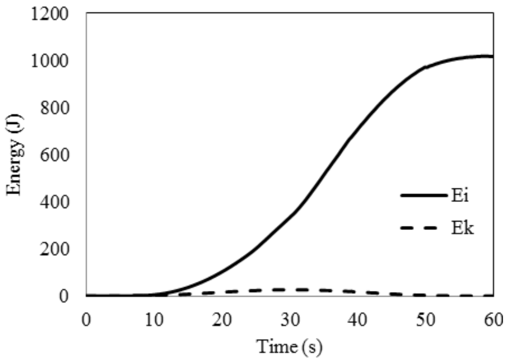


Fig. 6 (c) Energy-time relationship with LR=0.5 and MSF=10^6

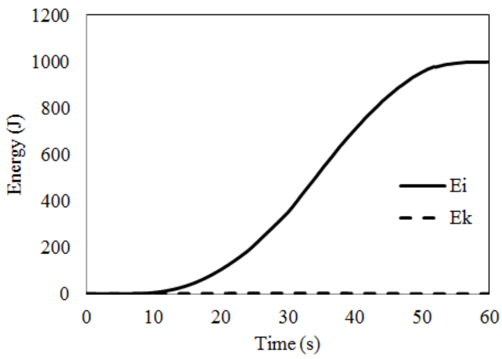


Fig. 6 (d) Energy-time relationship with LR=0.5 and MSF=10^5

After selecting the initial values of loading rate and mass scaling factor, the model should be validated with the experimental results in terms of load-displacement relationship, strain-displacement relationship and the deformed shape. The comparison was made and good agreement was achieved for one tested sample. The other samples were also then validated using the same loading rate and mass scaling factor with good agreement. Fig. 7 shows the verification of the load displacement curve of the numerical model and the experimental result of one sample. It is clear that the model is capable to predict the bilinear behaviour of the connection under quasi-static load, in spite of the noise that could appear in the late stages of the analysis. This noise could be avoided using lower loading rate and mass scaling factor, which in turn leads to an increase in the CPU time. For validation purposes, the strain at four different units was measured up to the maximum displacement and all results were correlated to the numerical results. Also, the experimental failure mode of the tested samples shows very good correlation to that in the FE models, as shown in Fig. 8.

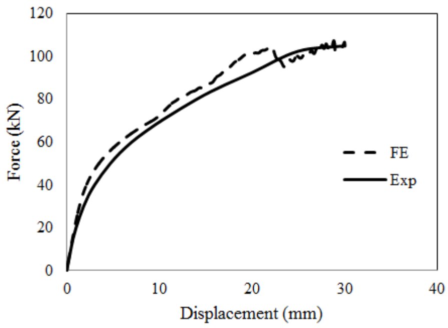


Fig. 7 Validation of Load-displacement curve

It should be mentioned that an analysis was performed using the selected loading rate (i.e. 0.5 mm/s) with no mass scaling factor. The results show very good correlation to the experimental results and the numerical results using the same loading rate with a mass scaling factor of 10^6. However, the CPU time of the analysis with no mass scaling factor was about 38 times that without this factor.

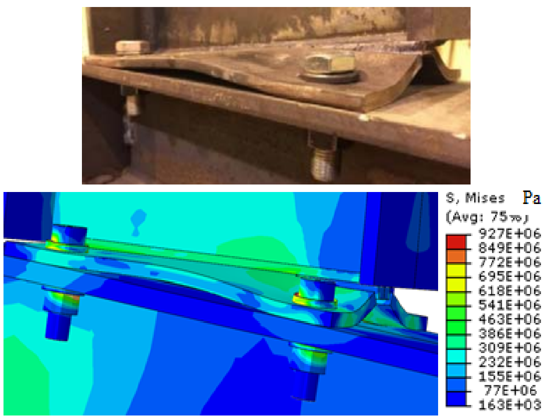


Fig. 8 Validation of the deformed shape

IV. Conclusions

This paper reported on the development of a finite element model using explicit procedure to estimate the behaviour of bolted end plate connections under quasi-static loading. The models were validated with four tested specimens in terms of load-displacement relationships, strain-displacement relationships in critical points and deformed shapes. Good agreement has been achieved and the explicit solution technique can be classified as a reliable tool to effectively simulate the behaviour of such connections under quasi-static loading. Also, it is concluded that a combination of low loading rate and a mass scaling factor of 106 could be used to minimise the CPU time and to eliminate the dynamic effect that may be produced using explicit technique.

Acknowledgment

The work presented in this paper was supported by the Ministry of Higher Education and Scientific Research in Iraq, which is gratefully appreciated. The authors wish to thank Mr. J. Curran and Mr. G. Friel for their assistance in the experimental work.

References

[1] Way A. G., Structural robustness of steel framed buildings. 2011: SCI. 55.W.-K.
[2] Krishnamurthy N. and Graddy D. E., *Correlation between 2- and 3-Dimensional Finite Element Analysis of Steel Bolted End-Plate Connections.* Computers and Structures, 1976. 6: p. 381-389.
[3] Kukreti A. R., Murray T. M., and Abolmaali A., End-Plate Connection Moment-Rotation Relationship. Journal of Constructional Steel Research, 1987. 8: p. 137-157.
[4] Vegte G. J., Makino Y., and Sakimoto T., *Numerical Research on Single Bolted Connections Using Implicit and Explicit Solution Techniques.* Memoirs of the Faculty of Engineering, Kumamato University, 2002. 47.
[5] Bursi O. S. and Jaspart J. P., *Benchmarks for Finite Element Modelling of Bolted Steel Connections.* Journal of Constructional Steel Research, 1997. 43: p. 17-42.
[6] Bursi O. S. and Jaspart J. P., Calibration of a Finite Element Model for Isolated Bolted End-Plate Steel Connection. Journal of Constructional Steel Research, 1997. 44: p. 225-262.
[7] Wheeler A. T., Clarcke M. J., and Hancock G. J., FE Modelling of Four Bolt Tubular Moment End-Plate Connections. Journal of Structural Engineering, 2000. 126: p. 816-822.
[8] Abdullah R., Paton-Cole V. P., and Easterling W. S., Quasi-static analysis of composite slab. Malysian Journal of Civil Engineering, 2007. 19(2): p. 91-103.
[9] Natário P., Silvestre N., and Camotim D., Web crippling failure using quasi-static FE models. Thin-Walled Structures, 2014. 84: p. 34-49.
[10] Abaqus documontations. SIMULIA.