International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 5.22 (SJIF-2017), e-ISSN: 2455-2585
Volume 5, Issue 04, April-2019

Finite Element Analysis of Multi-storey Steel Frame Structure Subjected to Seismic Ground Motions: A Review

Bisakha Chalisey[1], Bikram Raj Gyawali[2], Megha Gupta[3]

[1]*Department of Civil Engineering, Sharda University,*
[2]*Department of Civil Engineering, Sharda University,*
[3]*Department of Civil Engineering, Sharda University,*

Abstract*— Finite element analysis (FEA) is a very popular method of analysis whose popularity is increasing day by day in the design of steel structures in order to avoid unsafe or highly conservative designs. For a Structural Engineer, to perform a finite element analysis, solid know-how in computer-aided design (CAD) and engineering (CAE) codes is necessary. The paper reviews the detailed finite element analysis of steel frame structure subjected to consecutive ground excitations conducted using E-Simulator. E-Simulator is a parallel finite element analysis software package developed to accurately simulate structural behaviour up to collapse by using a fine mesh of solid elements. Therefore the purpose of this paper is to provide an extensive review of useful guidelines concerning modelling, simulation and result validation for the accurate performance of finite element analyses.

Keywords- Finite element analysis, E-simulator, Consecutive excitation, modelling, computer-aided design.

I. INTRODUCTION

The finite element method has become a powerful tool for the numerical solution of a wide range of engineering problems. Applications range from deformation and stress analysis of automotive, aircraft, building, and bridge structures to field analysis of beat flux, fluid flow, magnetic flux, seepage, and other flow problems with the advances in computer technology and CAD systems, complex problems can be modelled with relative ease. Several alternative configurations can be tested on a computer before the first prototype is built while we are talking about Multi-story steel structure; it has made the design in precise and accurate manner regarding dynamic response.

It is a well-known fact that structures are subjected to three different dimensional earthquake ground motions. But earthquake design (ELF Method) for buildings usually only considers the horizontal component of ground motion while the vertical component of ground motion is generally neglected. Many building codes including NBC 105, IS 1893, UBC 97 and many other codes worldwide assume the vertical component of the ground motion to be 2/3 of the horizontal component which is originally provided by Newmark et al (1973). Figure 1 shows the ground motion acceleration records of E1-centro 1940 (Shrestha, 2009).

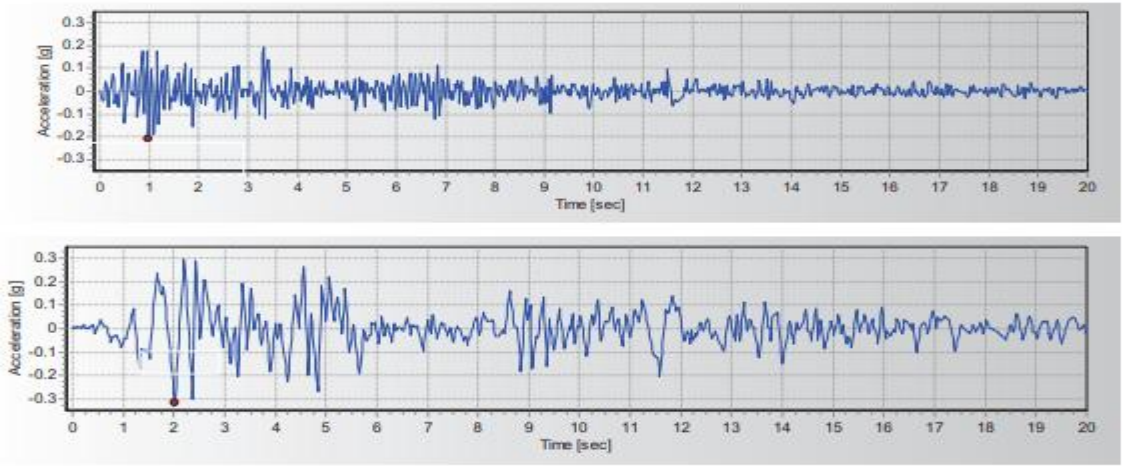

Fig 1 Ground motion timehistory of El-centro 1940 vertical(0.21g) and horizontal(0.32g) respectively from top

Fig. 1: Ground Motion Time History of El-Centro 1940 Vertical (0.21g) and Horizontal (0.32g) Respectively from Top [Shrestha, (2009)]

This type of dynamic response encounter while designing so during preliminary design of skyscraper nowadays engineer prefer the steel structure rather than RCC because of low cost in return period, ductility is high, can be reuse and so no. While designing of multi-storey steel building software which include algorithm of FEM is consider to be the best because complex region defining a continuum is discredited into simple geometric shapes called finite elements. The material properties and the governing relationships are considered over these elements and expressed in terms of unknown values at element comers. An assembly process, duly considering the loading and constraints, results in a set of equations. Solution of these equations gives us the approximate behaviour of the continuum.

II. LITERATURE REVIEW

Mizushima *et. al,* (2017) conducted an experimental study based on numerical analyses that are conducted for a steel structure subjected to multiple series of excitations in a full-scale shaking-table test considering fractures. The structure was modelled with planar and solid finite elements and the fracture is treated by the mandatory deletion of elements at the time at which the fracture is observed in the experiment. The FE model was elaborated to replicate adequately entire shape of the specimen. All steel members were modelled with first-order planer elements that have 4-node at a unit length of 25 mm except for columns on the lines A and C. The planer elements had 4 integration points in the thickness direction and 1 integration point in the plane. The scallops were made to become single arc, whose radius was 35 mm, following the conventional approach used at the time of the Hyogoken-Nambu earthquake. The results show that by considering the fracture of steel members with the deletion of elements, the history of input excitations, and the resulting damages, the behaviours can be simulated analytically with a much higher accuracy.

Tagawa *et. al,* (2015) conducted an experimental investigation to study the detailed finite element analysis of a full-scale four storey steel frame structure subjected to consecutive 60% and 100% excitations from the JR Takatori records during the 1995 Hyogoken-Nanbu earthquake using E-simulator. An FE model for the four-storey steel frame structure, which primarily consists of hexahedral solid elements has 4,543,243 elements, 6,330,752 nodes, and 18,992,256 degrees of freedom. The result showed that the first-storey drift response for the consecutive 100% excitation after 60% excitation was slightly smaller than that for single 100% excitation. It was also found out that complete collapse which occurred in the E-Defense shake table test, was not observed by the analysis for the consecutive 60% and 100% excitations or for the single 100% excitation.

Chik *et. al,* (2015) investigated the natural frequency and vibration mode of multi storey office building with the presence of foundation system and comparison between both systems. Finite element modelling (FEM) package software of LUSAS is used to perform the vibration analysis of the building. The building is modelled based on the original plan with the foundation system on the structure model. The result showed that the structure which modelled with rigid base have high natural frequency compare to the structure with foundation system. These maybe due to soil structure interaction and also the damping of the system which related to the amount of energy dissipated through the foundation soil.

Miyamura *et. al,* (2014) performed dynamic finite element analysis of a four-storeyed steel building frame modelled as a fine mesh of solid elements, performed using E-Simulator which is a parallel finite element analysis software package for precisely simulating collapse behaviours of civil and building structures. A mesh of the entire structure of a four-story frame with approximately 19 million degrees of freedom is constructed using solid elements. The density of the mesh is determined by referring to the results of elastic–plastic buckling analyses of a column of the frame using meshes of different densities. Seismic response analyses under 60, 100, and 115% excitations of the JR Takatori record of the 1995 Hyogoken-Nanbu earthquake are performed. It can be found that in the analysis under the 60% Takatori wave, the time histories of the inter-story drift angle and the story shear force obtained by E-Simulator show good agreement with the experimental results. Also in the analysis under the 100 and 115% Takatori waves, both the response of the entire frame and the local deformation as a result of plastic buckling are well simulated by E-Simulator. Because contact occurs between the column and slab, simulations using the models with and without the stick condition are performed.

Miyamura *et. al,* (2011) conducted an experimental study to assess finite element analysis of a full scale 4 storey steel frame and a 31-story super-high-rise steel frame. The 4-story frame is a specimen of the full-scale total collapse shaking-table test conducted in 2007. These steel frames are modelled by meshes of hexahedral solid elements. All the members and the floor slabs are modelled by 8-node hexahedral solid elements; i.e., the DOFs of each node correspond to three translational displacements, and the displacements in the elements are interpolated by linear shape functions. A 31-story super-high-rise steel building frame has been designed as a specimen for the E-simulator. The frame is a centre-core-type 31-story office building. The result shows that elastoplastic dynamic responses can be estimated with good accuracy using high-precision FE analysis without resorting to macro models such as those involving plastic hinge and composite beam effects.

Feng Fu (2010) conducted a study on a 3-D finite element model to analyse the progressive collapse of a multi-storey steel composite frame building. The behaviour of the 20 storey steel composite frame building under the sudden column removal was investigated with a 3-D finite element model using the ABAQUS package. Based on this model, parametric studies were carried out to investigate the structural behaviour with variations in the strength of concrete, structural steel and reinforcement mesh size. The findings were that the plasticity is presumed to develop in the steel member under the one column removal scenario, and a plastic hinge is formed in the beam. Therefore most research was based on the plasticity theory.

Oshaki *et. al,* (2009) conducted an experimental analysis of dynamic collapse behaviour of steel frames using large-scale parallel finite element method based on the domain decomposition method. The analysis software is based on ADVC as a part of the E-Simulator that takes advantage of the recent development of computer science and high-performance parallel computing in computational mechanics. Numerical results are shown for dynamic collapse analysis of single-story and 5-story frame models to show that the global and local behaviours are simultaneously simulated by a high-precision finite element analysis. Eigenvalue analysis is also carried out for a 31-story frame to demonstrate that dynamic analysis can be carried out for a structure discretized to solid elements with more than 70 million DOFs.

Liu *et. al,* (2008) conducted a detailed case study on structural identification and Finite Element (FE) modelling of a 14-story office building with system identification tools to identify the structural dynamic properties based on recorded seismic data. This identified model was compared with a model based on IBC 2000 (International Building Code) using the fundamental period, design response spectral acceleration, and base shear. It was found that the fundamental period of the building computed using the approximation formula in IBC 2000 was 20.8% shorter than the identified one, while the fundamental period calculated by FE modelling (Model 1) which considered generally accepted engineering practices and assumptions suggested by IBC 2000 was 33.5% longer than the identified as longer one.

III. CONCLUSIONS

Based on the study done, it can be concluded that the detailed finite element analysis of steel frame structure subjected to consecutive ground excitations, conducted using various techniques including E-Simulator, ABAQUS, LUSAS amongst others. The present study illustrates the feasibility of a large-scale structural analysis in the field of earthquake engineering and is novel from the viewpoint of computational mechanics. A number of studies have achieved large-scale highly nonlinear finite element structural analysis using a mesh with approximately 20 million DOFs, even in other fields, such as mechanical engineering.

REFERENCES

[1] B.Shrestha, “ Vertical ground motion and its effect on engineering structure: state of the art review”, International Seminar on Hazard Management for Sustainable Development November 29-30 2009 ,pp.190-202

[2] Tagawa H., Miyamura T., Yamashita T, Kohiyama M, Oshaki M., (2015). “Detailed Finite Element Analysis of a Full scale Four storey Steel Frame Strcuture” International Journal of High rise Buildings Number 1

[3] Mizushima, Yasunori & Mukai, Yoichi & Namba, Hisashi & Taga, Kenzo & Saruwatari, Tomoharu. (2018). Super-detailed FEM simulations for full-scale steel structure with fatal rupture at joints between members-Shaking-table test of full-scale steel frame structure to estimate influence of cumulative damage by multiple strong motion: Part 1. Japan Architectural Review. 10.1002/2475-8876.10016.

[4] Tuan Chik, Tuan Norhayati & F Zakaria, M & A Remali, M & Nor Azizi, Yusoff. (2016). Vibration Response of Multi Storey Building Using Finite Element Modelling. IOP Conference Series: Materials Science and Engineering. 136. 012037. 10.1088/1757-899X/136/1/012037.

[5] Miyamura, Tomoshi & Yamashita, Takuzo & Akiba, Hiroshi & Ohsaki, Makoto. (2014). Dynamic FE simulation of four-story steel frame modeled by solid elements and its validation using results of full-scale shake-table test. Earthquake Engineering & Structural Dynamics. 44. 10.1002/eqe.2526.

[6] Miyamura, Tomoshi & Ohsaki, Makoto & Kohiyama, Masayuki & Isobe, Daigoro & Onda, Kunizo & Akiba, Hiroshi & Hori, Muneo & Kajiwara, Koich & INE, Tatsuhiko. (2011). Large-Scale FE Analysis of Steel Building Frames Using E-Simulator. Progress in Nuclear Science and Technology. 2. 651-656. 10.15669/pnst.2.651.

[7] Fu, Feng. (2010). 3-D nonlinear dynamic progressive collapse analysis of multi-storey steel composite frame buildings — Parametric study. Engineering Structures. 32. 3974-3980. 10.1016/j.engstruct.2010.09.008.

[8] Ohsaki, Makoto & Miyamura, Tomoshi & Kohiyama, Masayuki & Hori, Muneo & Noguchi, Hirohisa & Akiba, Hiroshi & Kajiwara, Koichi & Ine, Tatsuhiko. (2009). High-Precision Finite Element Analysis of Elastoplastic Dynamic Responses of Super-Highrise Steel Frames. Earthquake Engineering & Structural Dynamics. 38. 10.1002/eqe.900.

[9] Liu, He & Yang, Zhaohui & S. Gaulke, Michael. (2005). Structural identification and finite element modeling of a 14-story office building using recorded data. Engineering Structures. 27. 463-473. 10.1016/j.engstruct.2004.11.012.