SDSS'Rio 2010 STABILITY AND DUCTILITY OF STEEL STRUCTURES
E. Batista, P. Vellasco, L. de Lima (Eds.)
Rio de Janeiro, Brazil, September 8 - 10, 2010

NUMERICAL MODELING OF JOINT DUCTILITY IN STEEL AND STEEL-CONCRETE COMPOSITE FRAMES

Leslaw Kwasniewski and Marian Gizejowski

Warsaw University of Technology
e-mails: l.kwasniewski@il.pw.edu.pl, m.gizejowski@ il.pw.edu.pl

Keywords: Joints, Progressive collapse, Robustness, Numerical modeling, Verification and Validation.

Abstract. *The paper presents a study on numerical modeling of typical joints in steel and composite steel-concrete frame structures for the purpose of the global progressive collapse analysis of multistory buildings. The difficulty in such analyses is to efficiently model the response of joints with consideration of all the aspects of structural detailing, like the postwelding properties of steel, the weld size effect, detailed bolt connection geometry including the bolt head, nut and washer, as well as to merge responses of two materials into one representation in case of composite joints. The problem is investigated using nonlinear dynamic computer simulations carried out using general purpose program LS-DYNA. The feasibility study is focused on identification of modeling parameters affecting the final results and on development of a plan for hierarchical verification and validation.*

1 INTRODUCTION

In progressive collapse analysis of structures, the primary objective is to limit structural damage and in this way save lives, prevent injury, and protect property. Different design approaches have been developed for reducing or eliminating disproportionate damage. There have been attempts to design stronger members to resist specific abnormal loads or to limit the total damage by effective redistribution of loads, i.e., alternate load paths [1]. For all approaches, the principal question for structural analysis is the extent of damage caused by a local failure initiated by an infrequent event. In the most common progressive collapse analysis, the usual loading procedure is the removal of one or more columns [1]. To find the extent of damage caused by a local component failure, the whole structure or a large portion of it has to be analyzed. Such analysis is rather complex as actual structural collapse itself is inseparably a nonlinear event in which structural elements are stressed beyond their elastic limit up to failure. There is also a need for global modeling, where large portions of a building are represented but at the same time capturing local effects is equally important. The possible load paths are dependent on the performance of localized plastic zones. To introduce plastic hinges in the analysis, generally two approaches can be applied. In the first, simplified approach, plastic hinges with the assumed characteristics in terms of moment–rotation relationships are introduced explicitly in the model. Such moment-rotation relationships are usually determined for planar bending only. Structural components such as beams and columns are usually represented by 3D beam elements in these models. The outcome of such analyses is strongly determined by the assumptions made.

The paper presents a study on more detailed numerical modeling of typical joints in steel and composite steel-concrete frame structures. The main objective of this research is to efficiently model beam-to-column joints for the purpose of the global progressive collapse analysis of multistory buildings. Structural joints, such as the considered here steel and composite steel-concrete joints are challenge for numerical modeling, especially when the full range of loading up to failure is considered. The difficulty is to efficiently model the response of joints with consideration of all the aspects of structural detailing, like the post-welding properties of steel, the weld size effect, detailed bolt connection geometry including the bolt head, nut and washer, as well as to merge responses of two materials into one representation in case of composite joints. For small specimens of structural joints, the most accurate approach is to model all the joint details and each material separately, for example, the bolt connectors and nuts as well as a concrete slab core with solid elements, while the slab profiled deck and other joint components with shell elements, and reinforcement bars with truss or beam elements. For large-scale models this strategy is infeasible today as it would result in a very large number of finite elements.

The problem is investigated using nonlinear dynamic computer simulations carried out using general purpose program LS-DYNA. Taking advantage of parallel processing on multiprocessor computers, detailed 3D models with different numbers of finite elements, interactions and other solution parameters have been developed for comparison among numerical and experimental results. Using transient nonlinear dynamic simulations selected connections are tested numerically for full range of loading up to failure. Typical plane bending and loading initiated by notional column removal is considered. The feasibility study is focused on identification of modeling parameters affecting the final results and on development of a plan for hierarchical verification and validation.

2 DETAILED NUMERICAL MODELING OF STEEL JOINTS

2.1 Solution methods

The review of published works on progressive collapse analysis (compare bibliography in [2]) shows that there is a broad variety of approaches and one can choose between linear and nonlinear, static and dynamic, and between 2D and 3D analyses. The nonlinear time history (dynamic) analysis is often recognized as giving most realistic results but at the same time, due to its high complexity, it is prone to incorrect assumptions and modeling errors.

Depending on how time is treated in the analysis, we can choose among dynamic, quasi-static, and strictly static approaches. The loss of stability is usually a dynamic process, and therefore should be directly traced using the most general, dynamic approach. Nonlinearity and discontinuity cause convergence problems that are less severe in the dynamic due to the stabilizing effect of the inertia forces [3].

A dynamic analysis with the finite element (FE) method applied for space discretization is usually conducted using implicit or explicit time integrators. Implicit dynamic analysis (using implicit time integrators) is dedicated for structural problems described by Belytschko *et al.* [3] as inertial problems where stress wave propagation and related effects are not important. For such problems, the response time is relatively long compared to the time required for a stress wave to traverse the structure. When the response time sought is short and the wave effects are important, the time step must be very small, and more appropriate solution methods are those based on the explicit time integration.

The explicit time integration scheme belongs to purely incremental methods and is applicable to formally dynamic problems. The incremental solution methods dominate incremental-iterative methods, especially for problems experiencing rough nonlinearities, which involve component failure or inequality constraints such as contact or friction [3]. In the explicit methods, the equations of motion are usually solved using the central difference method with very small time steps determined by the highest frequency of the linearized system [4]. The codes based on the explicit time integration are dedicated to dynamic transient problems and have proved to be especially effective when large deformations grow rapidly. Contrary to the implicit methods, the explicit time integration cycle is computationally much less expensive as there are no iterations and only one diagonal matrix needs to be inverted. However, the simulated period of time requires a much larger number of calculation cycles. With a time step of the order of microseconds, even a few seconds of a typical simulated event require millions of calculation cycles and substantial calculation time. The explicit method is simple, easy to implement, and very effective; it rarely aborts due to failure of the numerical algorithm, which quite often is the problem for implicit methods.

2.2 Material properties

In steel and composite, steel–concrete, frame structures, working in the inelastic range, deformation is concentrated in plastic hinges and zones. The performance of the plastic hinges, usually localized at joints, depends strongly on the inelastic properties of the steel components [5]. The critical parameters are yield stress, ultimate stress, and ultimate strain. If the actual stress-strain relationships are measured through laboratory coupon tests, the input for a numerical model should be formulated as a true stress-strain curve, recalculated based on the directly measured engineering values and well known formulas [4]

$$\sigma_T = \sigma_E (1 + \varepsilon_E), \tag{1}$$

$$\varepsilon_T = \ln(1 + \varepsilon_E), \tag{2}$$

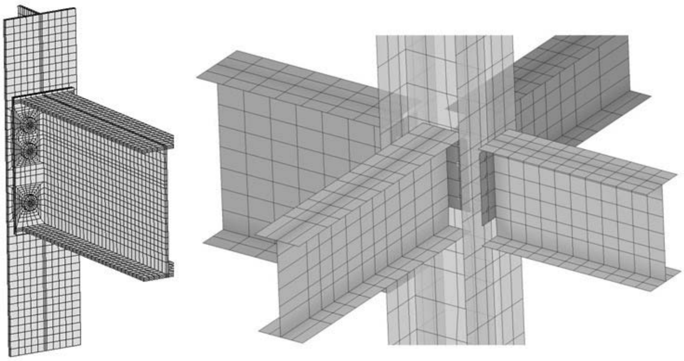
where subscripts $_T$ and $_E$ denote true and engineering values, respectively. If only nominal properties are known for the specific structural steel grade, simplified material models have to be applied, such as the elastic—ideally plastic or bilinear elastic—plastic model with hardening, recommended by Eurocode 3 [6]. Although progressive collapse is generally a dynamic event, happening in a matter of seconds, the strain rate effects seem to be not so important at least for the threat independent approach (but should be considered in the impact or explosion analyses).

Localized effects of the phase transformation and restrained shrinkage in post-welded steel components of joints are important for the failure assessment [7]. Welds of full penetration, the quality of which is guaranteed by non-destructive testing, are usually stronger-in-strength places than the surrounding parent material. The ductility requirement for welds is not important for the adequately selected weld consumable material, and failure of such connections may occur in the vicinity of welds, in the so-called heat affected zones (HAZ). Ductility and strength of steel in this region is controlled by the properties of different steel transformation phases such as hardened non-ductile martensite phase, then less hardened and more ductile bainite phase, and finally ductile pearlite phase of the parent material. Depending on the post-welding cooling process of steel from the austenite, different residual stress patterns and phase transformation components may exist in the region of HAZ and affect substantially the localized mechanical properties of steel, ability to plastic behavior and failure by cracks growth. Modeling of localized effects in the post-welded steel is not an easy task and usually neglected in bolted end-plate connections in which the ductility of high strength bolt material becomes the decisive failure criterion. The effect of residual stresses may be introduced by applying the concept of equivalent stress-strain diagram.

Concrete is at the same time a common construction material and the biggest challenge for numerical modeling. As a porous and brittle material, concrete shows a nonlinear compaction response, dramatically different strengths in tension and compression, and shear strength increased by mean stress [8]. Although in practice concrete is characterized by only one parameter, uniaxial unconfined compressive strength (f'_c), concrete's complex behavior under different stress states requires from a user determination of many parameters, stress-strain curves, and sometimes equations of state for pressure versus volume strain [4]. As such experimental data are usually not available, some of the new material models for concrete implemented in the commercial codes (e.g., LSDYNA) offer an option for automatic generation of all required input data. The internal generation is based on the well-worked laboratory experimental data for selected examples of standard concrete. The user is required only to provide unconfined compressive strength f'_c. The program provides generated parameters that can be modified and applied as the new input by a user who has additional information about the modeled concrete. Such an approach seems to be very convenient for users and sufficiently reliable for most engineering applications. This reliability is based here on the assumption that although there are many types of concrete standard materials with the same f'_c show similar properties for other stress states. In contrast to accuracy, which seems to be sufficient based on the single element tests [8], the main computational difficulty emerging here is the stability of the calculation, especially when the analysis is supposed to reach beyond the local material damage. To avoid numerical problems caused by extensive deformation, elements need to be erased in accordance with the prescribed failure criteria.

2.3 Space discretization

A modeling issue which should be considered here first is the space discretization allowing for adequate representation of component stiffness especially for connections where most of the inelastic deformation is concentrated. The space discretization for structural analysis is nowadays usually performed using finite elements (FE). Figure 1 shows two concepts for FE models of framed structures. In the first model (on the left) all components of the joint are modeled in very detailed way using solid elements. Such approach increases substantially a number of finite elements in the model but allows for capturing effects initiated by large structural deformations such as an inelastic bending of end plates or stress concentration in the vicinity of bolts. Due to large number of finite elements required, such models are infeasible for global modeling but can be used for partial verification of less detailed models such as one shown on the right in Figure 1. In this approach all beams and columns are represented by 3D FE models built of shell elements. The bolts are represented by 1D beam elements. Additionally, there is a global contact defined among all the steel components represented by shell elements, e.g., flush end plates and columns, and eventually shell elements representing floor slabs (not shown in Figure1). The contact is defined using an offset equal to half of the thickness for each interacting metal sheet.



a) b)

Figure 1: Modeling concepts for frame structure: a) 3D model built of solid elements, b) 3D model built of shell elements.

The model built of shell elements, shown in Figure 1 is a compromise between tendency of using detailed models and computational capabilities of today's computers.

3 VERIFICATION AND VALIDATION

The predictive capability of the nonlinear dynamic simulations is dependent on the inherent complexity of the method (e.g. contact and failure algorithms), taken assumptions (modeling simplifications) and uncertainties characterizing the input data (e.g. material parameters determining component failure). To improve the validity of the developed FE models and to identify the decisive model parameters a hierarchical verification and validation (V&V) procedure should be considered [9]. While verification uses analytical or highly accurate numerical solutions, validation is based on the comparison of computational results with experimental data. The hierarchy of the process means that the comparison is performed on different levels of complexity, for example from material characterization or element tests through subsystems to the full scale tests [9]. Many consider the V&V as a process which can never be completed as there are always some possible experiments left which could reduce errors of the solution and the available recourses are always limited. Practically the amount of experimental data which can be used for validation is always limited. A compromised solution is to replace some of the missing experimental data from the purposely designed validation tests (e.g. on joints) on the actual structure with numerical virtual tests using more detailed FE models of structural components. The emphasis should be put on the identification of the model parameters critical for the outcome of the computer simulations. This objective can be accomplished by showing the results for assumed bracket quantities of critical model parameters.

3.1 Mesh resolution

The main part of verification, which should precede validation, is the study on mesh resolution. Figure 2 shows as an example two of the FE meshes, called coarse and dense meshes, considered for an example beam to column connection. A 356x171x51UB (S355) beam is bolted on the flange of a column 305x305x137UC (S355) through an end plate (10 mm) with eight M20 grade 8.8 bolts. In this FE model the beam and the column, including flush plate are modeled using (first-order) shell elements, see Figure 1.

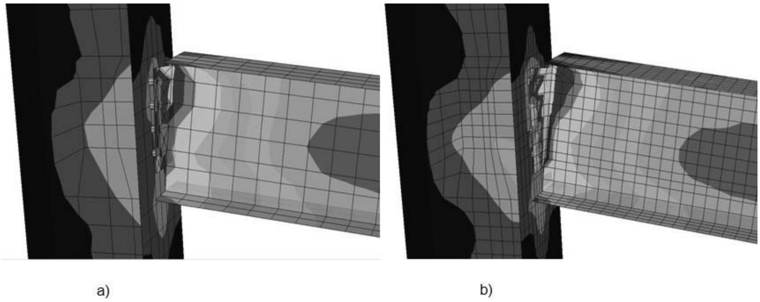


Figure 2: Study on mesh sensitivity for 3D FE model of joints: a) coarse mesh, b) dense mesh.

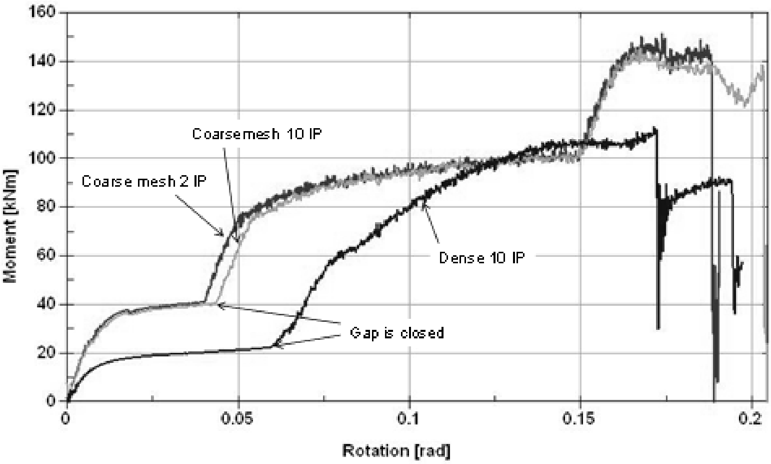


Figure 3. Comparison of moment-rotation curves for different FE models of beam to column joint.

Figure 3 presents comparison of calculated moment–rotation curves for typical tested joints with the same flush end-plate connections, i.e. in planar bending (with lateral movement restrained). The numerical results were obtained using dynamic analysis with explicit time integration. The curves shown in Figure 3 represent different mesh densities and different number of through thickness integration points (IP). Although the number of integration points has little effect on the results the finest mesh considered here gives twice smaller initial stiffness and 24% smaller approximation of the maximum bending moment in the reference to the coarse mesh. The initial stiffness of the connection is mostly determined by bending of the flush plate until the gap between the bottom beam's flange and the column's flange is closed and the visible in Figure 3 hardening phase begins. At the same time all models give similar approximation of the ultimate (failure) rotation. There is also clear difference between the results for coarse and dense FE meshes in terms of work done by loading as it shown in Figure 4.

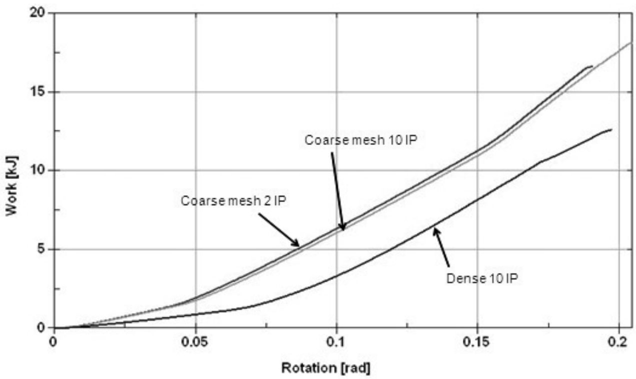


Figure 4. Comparison of work done by loading for different FE models of beam-to-column joint.

The dense mesh shown in Figure 2 might be inappropriate for global analyses as infeasible with available computational resources that are giving too small time steps and too large number of elements. This example illustrates a difficulty in finding good balance between the necessity of global analysis of large systems and requirements for the proper representation of some structural details, e.g., connections between components, to correctly capture the local effects, which in turn can determine the further solution.

3.2 Failure strain for structural steel and bolts

The empirical studies on beam-to-column joints indicate that the failure in the actual connections usually is initiated by rapture of fillet welds or failure of bolts (e.g. shear stripping of the threads), refer to [5]. In the FE models the material failure leading to the component disintegration is represented by deletion of a finite element from further calculations. For structural steel the most common failure criterion, determining an element deletion, is defined by a limit value of the effective plastic strain. Figure 5 shows comparison of three curves for the dense mesh and assumed different failure strains. Curve 1 was obtained for failure strain for bolts and structural steel ε_f =0.03, curve 2 for failure strains for bolts and flush plate increased by 20%, and curve 3 for bolt's failure strain increased by 43%.The coupon tests presented in [10] show high variation of the ultimate strains for the structural steel with the minimum magnitudes reaching almost half of the mean value.

Figs 6 and 7 present the results of the computer simulations for a major axis connection tested experimentally by Aribert *et al.* [5]. A HEA 360 (S355) beam is bolted on the flange of a column HEA 240 (S355) through a partial end plate (12 mm) with four M20 grade 8.8 bolts. This is a type of joint particularly convenient to use for a fast structure erection and although designed to transmit no bending moments, is actually able to carry a partial bending moment. Figure 6 shows the contours of effective Mises stress obtained for the used model with mesh density comparable to the dense mesh shown in Figure 2. The diagram shown in Figure 7 presents a comparison of several calculated moment-rotation curves with the experimental data provided in [5].

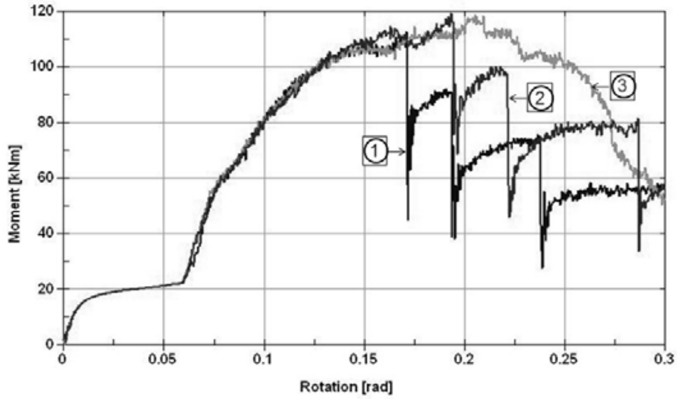


Figure 5. Comparison of moment rotation curves for different failure strains. Curve 1 – failure strain for bolts and structural steel $\varepsilon_f = 0.03$, curve 2 – failure strains for bolts and flush plate increased by 20%, curve 3 – bolt's failure strain increased by 43%.

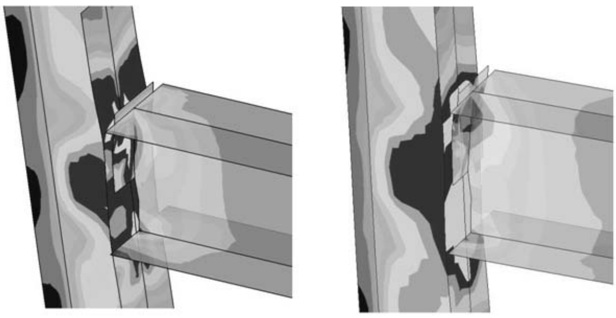


Figure 6. Contours of effective Huber-von-Mises stress calculated for maximum bending moments. Flush end-plate connection tested in [5] with different bolt's failure strain a) ε_f=0.02, b) ε_f=0.06.

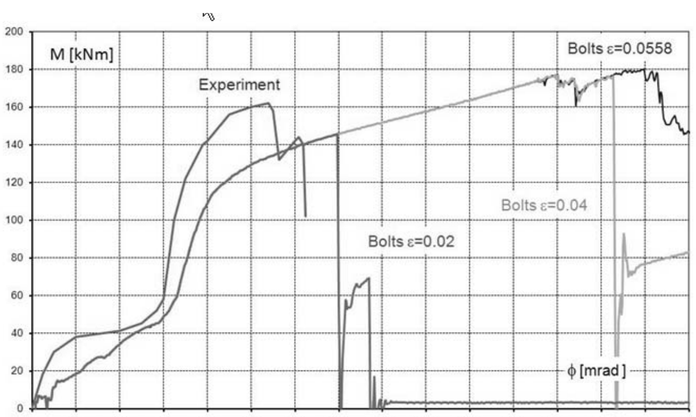


Figure 7. Comparison of experimental and numerical (coarse and fine meshes) moment-rotation curves.

The calculated curves in Figure 7 were obtained for different magnitudes of failure strain for bolts. The failure of the actual tested connection was due to a shear rupture of the nut threads of two bolts before ultimate tensile force (and failure strain) for the bolts were reached. To capture this effect in the FE model it is necessary to reduce the bolt's failure strain ε_f and treat it as a modeling parameter rather than as material characteristics. The comparison of moment-rotation curves shows that the FE model underestimates the stiffness of the connation, especially initial stiffness related to bending of the partial-depth end-plate. However, the simplified FE model is able to capture the clear increase in moment resistance when a contact between compression beam flange and column flange occurs and the center of rotation moves to a new location, the same effect as reflected in Figure 3.

4 CONCLUSION

The validation procedure presented here shows that the bolt's failure strain is the main parameter determining the moment and rotation capacities for beam to column connections and in this way one of the main modeling parameters affecting global analyses. Overall the most important modeling parameters which can potentially influence the performance of a joint are mesh density, especially for the flush end plates, failure strain for bolts, and the contact algorithm. The beam-to-column joints are usually tested experimentally for planar bending with transverse movements constrained. However, during collapse of a framed structure more complex loading configurations, with biaxial bending and torsion, are usually also present. Additionally a structural as a collapse can cause locally reverse loading, the correct representation of unloading in the constitutive relationships defining material models is also important.

REFERENCES

[1] General Services Administration (GSA), "Progressive collapse analysis and design guidelines for new federal office buildings and major modernization projects", Washington (DC) Office of Chief Architect; 2000.

[2] Kwasniewski, L., "Nonlinear dynamic simulations of progressive collapse for a multistory building", *Engineering Structures*, In Press, Corrected Proof, Available online 13 January 2010.

[3] Belytschko, T., Liu, W.K., Moran, B., "Nonlinear Finite Elements for Continua and Structures", *John Wiley & Sons, LTD,* Chichester, England.

[4] Hallquist, J., "LS-DYNA keyword user's manual", *Livermore Software Technology Corporation*, 2006.

[5] Aribert, J-M., Braham, M., Lachal, A., "Testing of ''simple'' joints and their characterisation for structural analysis", *Journal of Constructional Steel Research,* **60**, 659–681, 2004.

[6] ENV 1993-1-1:1992." Eurocode 3. Design of steel structures. General rules and rules for buildings", 1993.

[7] Gizejowski, M.A., Salah, W., Barcewicz, W., "Finite element modelling of the behaviour of steel end-plate beam-to-column joints". *Archives of Civil Engineering*, **54**(4), 693-733, 2008.

[8] Schwer, L.E., "Simplified concrete modeling with *MAT_CONCRET_DAMAGE_REL3", *LS-DYNA Anwenderforum*, Bamberg, 2005.

[9] Kwasniewski, L., "On practical problems with verification and validation of computational models", *Archives of Civil Engineering*, **55**(3), 323-346, 2009.

[10] da Silva, L.S., Rebelo, C., Nethercot, D., Marquesa L., Simõesa, R., Vila Real, P.M.M., "Statistical evaluation of the lateral-torsional buckling resistance of steel Ibeams", Part 2: Variability of steel properties. *Journal of Constructional Steel Research,* **65**, 832-849, 2009.