The Applied Element Method

the ultimate analysis of progressive collapse

By Hatem Tagel-Din, Ph.D. and Nabil A. Rahman, Ph.D., P.E.

Progressive collapse simulation is the latest challenge facing today's engineers wishing to assess the integrity of structures and to develop any necessary progressive collapse mitigation strategies.

To this end, an ideal progressive collapse numerical simulation should include the involvement and integration of the following elements: a modeling of the structural, non-structural components, and the load threat, the separation of parts, as well as the possible collision resulting from falling debris.

In numerical simulations, it is widely-recognized that the modeling of complicated physical systems often requires more time on preparation than in performing the actual computation of results. Since the modeling process depends on the analytical method used, it is important for engineers to select the proper analysis method that fits the complexity and time requirements necessary to create the model. Simplicity of structural modeling now becomes a key point in order to save time for revision and to assure a reliable degree of accuracy.

This article compares the modeling options offered via a new structural analysis technique for progressive collapse simulation, the Applied Element Method (AEM), with those of the current standard analysis method, the Finite Element Method (FEM).

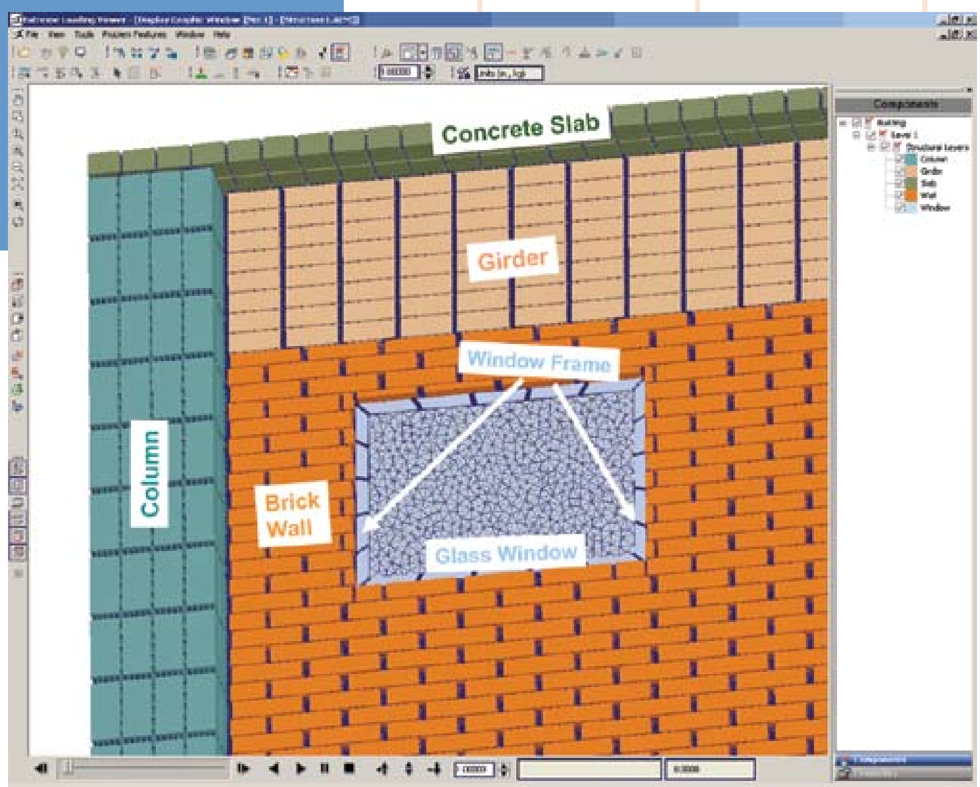

Figure 1: Simple structure

A Practical Example

For the purpose of argument, assume the simple structure shown in *Figure 1* is to be simulated under an extreme loading event, such as a bomb blast. The structure is composed of two reinforced concrete columns supporting a concrete girder and a roof slab, a brick wall, and glass window with an aluminum window frame. The nonlinear behavior of this simple structure must be simulated to evaluate the amount of damage that may occur to the main supporting elements, the wall and the glass window. Although this problem is not difficult from an analysis point of view, it is extremely complex from a modeling point of view when FEM is used.

Why Is It Complex to Use FEM for Such Modeling?

Modeling of 3-D structures using 3-D block elements with FEM is complicated and time consuming. Any FEM mesh has to represent the structural geometry and relationships between different structural elements through element connectivity. Most importantly, for progressive collapse simulations, it must be able to represent large displacements, stress and strain time history, contact and separation.

But, the main reason that the modeling of this behavior using the FEM may be complicated is because 3-D block elements should be used for all structural components to enable element contact while

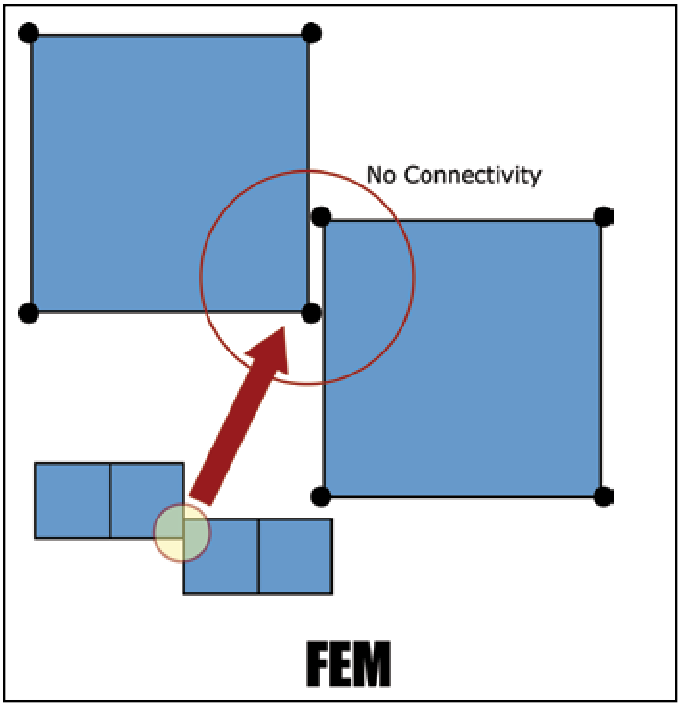

Figure 2: No connectivity. Node incompatibility is not allowed in FEM

still maintaining node compatibility between adjacent elements. As suggested in *Figure 2*, node incompatibility is not allowed in FEM. All elements should meet at corner nodes, otherwise the two elements are not connected and they behave like a crack in the model. To overcome this problem, transition elements are necessary in order to connect different mesh densities, as shown in *Figure 3*. In general, FEM transition elements include the following characteristics:

- They significantly increase the number of elements in the analytical model. (This can significantly slow down the computation of the analysis.)
- They may be automatically meshed. (Unfortunately, most automatic meshing tools use tetrahedral elements, which in many cases violate other meshing rules of the FEM, such as aspect ratio. Additionally there are still problems in automatically generating meshes with hexahedral shapes.)

Although there are other methods to connect adjacent joints, such as creating constraints between joints, they are still manually made and cannot be applied when elements are naturally staggered, as in the case of bricks.

Having established the limitations of the FEM mesh, let us return to the problem shown in *Figure 1*. The FEM mesh should be easy when modeling each component independently from the other components. But, a standard 3-D FEM mesh will still be very difficult to apply between:

- Staggered bricks and mortar
- Bricks and girder
- Bricks and column
- Bricks and window frame
- Window frame and glass

As a result, whether generated automatically or manually, the interference condition must be defined by the user on how to best model the above interfaces. By the use of transition elements shown at *Figure 3*, the mesh may require significant refinement, thus increasing the potential for errors in the analytical results.

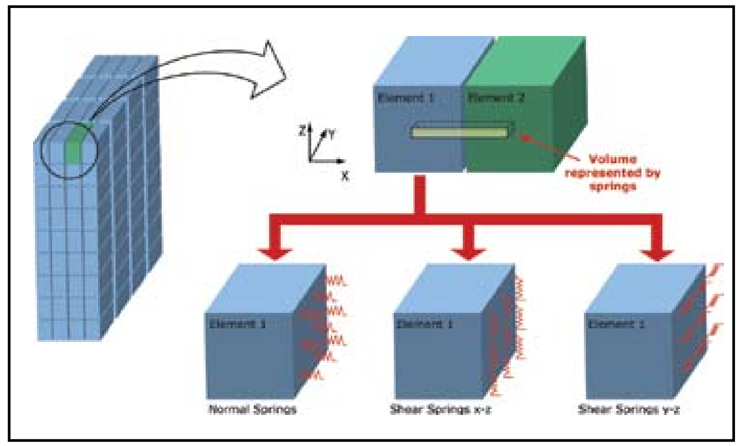

Figure 4: Connectivity of matrix springs

AEM Simplifies the Modeling of Complicated Geometries

The Applied Element Method (AEM) combines the advantages of FEM with that of the Discrete Element Method (DEM) in terms of accurately modeling a deformable continuum of discrete materials while maintaining an easy transition when material separates from continuum into a discrete component, which is needed when considering the debris field from blast loading or when a structure collapses.

continued on next page

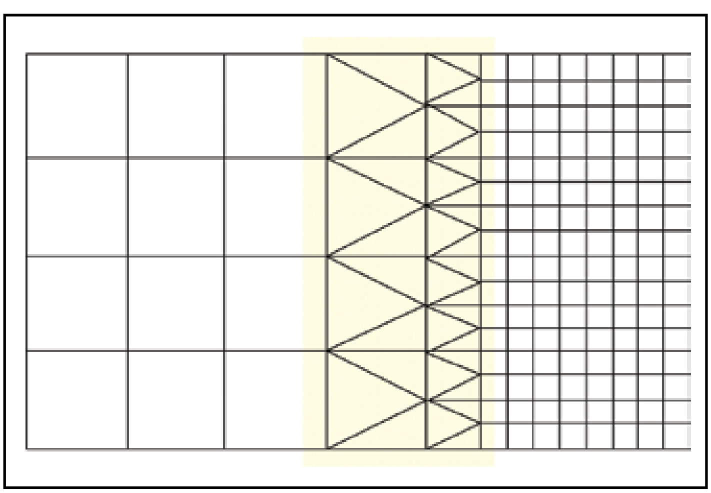

Figure 3: Use of transition elements at interfaces in FEM

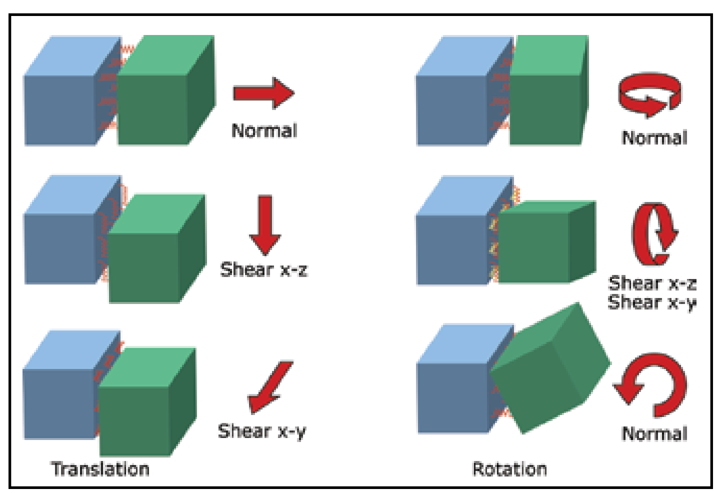

Figure 5: Degrees of freedom in AEM

Attribute	Finite Element Method (FEM)	Applied Element Method (AEM)
Connectivity	Nodes (Joints)	Element faces
Partial Connectivity	Not allowed, as nodes should be shared between elements	Allowed, as elements can meet at part of the face
Connections	Need transition elements	Not needed
Reinforcement	Special elements or embedded at Gauss Points	Connecting springs
Cracks in Brittle Materials	Difficult to localize, especially for smeared cracks	No smearing required
Element Separation	Elements can not be separated at their boundaries	Element separation at boundaries occurs and is automated

Table 1: Structural modeling in FEM vs. AEM

With AEM, the structure is modeled as an assembly of small elements, as shown in *Figure 4*. The two elements shown in the figure are assumed to be connected by a series of points. Each point has one normal and two shear springs, which are distributed around the element surface. Each element incorporates 3-D physical coordinates and shapes. Cuboids are used to model all structural components to be analyzed. The connecting springs may represent the reinforcement bars, the interface material or any desired material characteristics. Nonlinear material models are applied for the normal and shear springs. These material models account for, but are not limited to, the nonlinear hysteretic relations for steel and concrete in tension, compression and shear.

Six degrees of freedom are applied to each element, as shown in *Figure 5* which clearly illustrates that the moments and torsion resistance are a direct resultant of the resistance of individual springs around each element face.

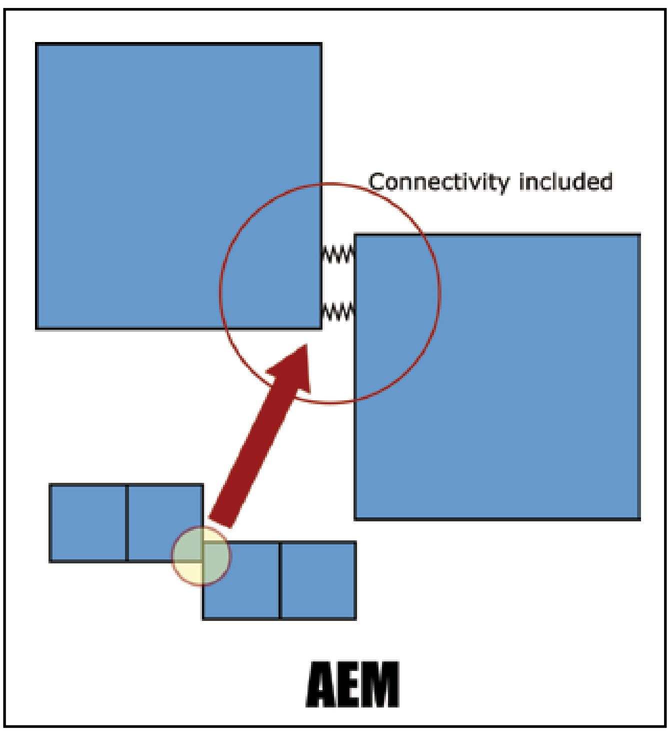

Figure 6: Connectivity included. Partical connectivity is allowed in AEM

What sets AEM apart from FEM in modeling is the way elements are connected together. Partial element connectivity (or node incompatibility) is allowed and springs are generated at the interface between elements (*Figure 6*). This eliminates the need for the transition elements, shown in *Figure 3*, and makes the meshing process easier to produce, as shown in *Figure 7*. In brief, by using the AEM there is no need for a complicated refined meshing of the continuum to accommodate the contact interfaces.

A reassessing of the complex scenario presented in *Figure 1* finds that all meshing problems are eliminated by simply allowing the partial element connectivity in AEM. Hence, connectivity springs are generated automatically throughout the interfaces between different objects. When engineers model such complicated structures, all that is required is to define the material type or property of different interfaces, instead of adding special elements or constraints at interfaces as in FEM. All interfaces are modeled without use of special elements or techniques, namely between individual bricks and mortar; bricks and girder; bricks and column; bricks and window frame and window frame and glass.

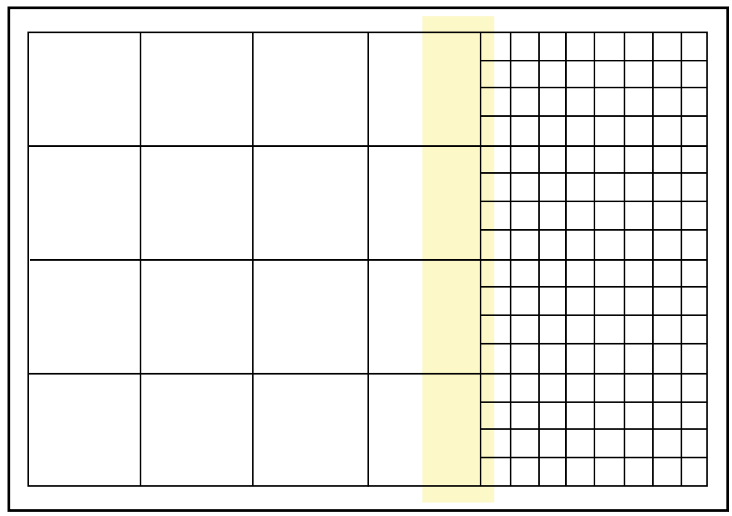

Figure 7: AEM does not require use of transition elements at connections

The Applied Element Method is not only superior to the Finite Element Method in terms of the modeling of structures; it also allows the user to obtain realistic results with progressive collapse simulations. Using AEM, element separation and motion as rigid bodies in the space contact with other elements is all automated without any user intervention. This means that not only modeling time is reduced, but easier and more accurate progressive collapse analysis can be performed. By simplifying the modeling process, even relatively inexperienced engineers can perform complicated simulations of progressive collapse without the extensive need for studying advanced FEM simulation methods, nonlinear material behavior or dynamic analysis. *Table 1* shows a comparison between the FEM and AEM for some of these features. *Figures 8a and 8b* displays AEM simulation results of the original situation subjected to blast pressure from a 100-pound TNT explosive source at 100 feet away from the wall, incorporating the fragmentation of the glass windows and the failure of the brick wall. Using AEM, the modeling of this problem was completed in five minutes and the initial results were obtained in fifteen minutes, using a personal computer.

Conclusions

The Applied Element Method provides a fast and accurate alternative to the Finite Element Method of analysis in the modeling and simulation of progressive collapse on structures. By including both structural and non-structural components, a truer representation of a progressive collapse scenario is generated and engineers, at last, have a tool available that allows them to determine the total effect of an extreme loading event.▪

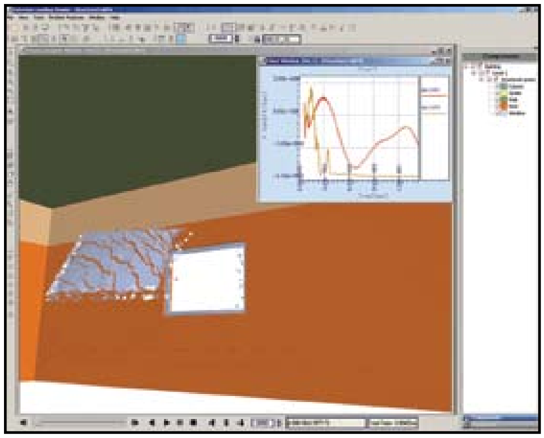

Figure 8a: Failure of the glass window

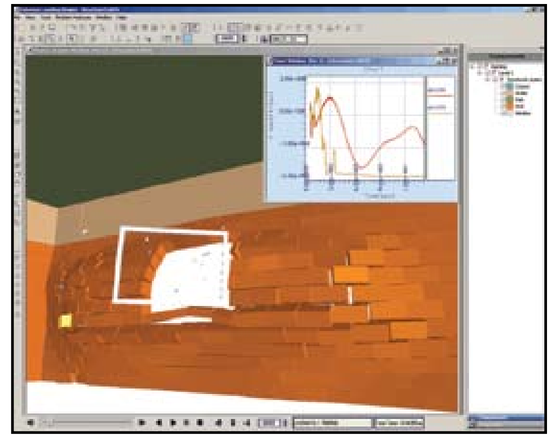

Figure 8b: Failure of the brick wall

*Nabil A. Rahman, Ph.D., P.E. is the Director of Research and Development for Applied Science International, LLC (ASI) and its parent company, The Steel Network, Inc. Dr. Rahman is a professional structural engineer with significant experience in numerical simulations, including both the Finite Element Method and the Applied Element Method. His experience also includes blast and progressive collapse analysis and design of structures, rehabilitation of structures, and light steel framing design. He currently serves as the Chairman of the Wall Stud Task Group of the Committee on Framing Standards (COFS) of the American Iron and Steel Institute (AISI). He is also a member of the ASCE-SEI Committee on Cold-Formed Steel as well as the AISI Committee on Specification. Dr. Rahman's can be reached via e-mail at **nabil@appliedscienceint.com***

*Hatem Tagel-Din, Ph.D. is the Project Manager of Extreme Loading for Structures software at Applied Science International, LLC (ASI). Dr. Tagel-Din is the founder of the Applied Element Method for structural analysis. Dr. Tagel-Din can be reached via e-mail at **hatem@appliedscienceint.com***