



# INSIGHTS

## FOR MASONRY DESIGN

### **Finite Element Modeling, Analysis, and Design for Masonry**

As structural analysis software evolves, the complexity and capabilities of these tools continue to expand, enabling engineers to enhance both the accuracy of structural assessments and the efficiency of the design process. Finite Element Analysis (FEA) has become indispensable in resolving complex structural problems, offering a method for approximating real-world behavior by discretizing a continuum with infinite degrees of freedom into a finite number of manageable elements. The application of FEA allows structural engineers to simulate stresses, deformations, and load distributions across masonry components to achieve safe and code-compliant designs.

#### **Introduction**

What exactly is finite element analysis? It is the process of reducing (simplifying) a problem with infinite degrees of freedom to a finite number of elements with unique material properties. FEA programs are able to resolve even the most complex of problems in a reasonable amount of time. The process of finite element modeling and analysis is an approximate solution which closely mimics an actual structure in a way that allows structural engineers to safely design for the stresses, forces, and deflections that are determined from these methods.

Some of the more commonly used software programs for FEA with masonry design are RAM Elements (soon to be released as STAAD(X) from Bentley Systems, Inc.) and RISA Floor/RISA 3D (from RISA Technologies). Other FEA programs with high-end analysis features, such as SCIA Engineer, are important tools for structural engineers because they offer more options for creating elements that more closely represent the actual elements behavior.

#### **General Comments about Finite Element Modeling**

Finite element models are created by modeling line, plate/shell, and solid (or brick) elements, with associated end nodes. The primary element types used in FEA for masonry are shown in Figure 1 and listed below:

**Line Elements:** Representing one-dimensional structural members, such as beams and columns, by defining axial, bending, and shear stiffness properties.

**Plate/Shell Elements:** Suitable for two-dimensional surfaces, these elements model slabs, walls, and shells, incorporating out-of-plane bending, in-plane shear, and axial stiffness characteristics.

**Solid (Brick) Elements:** Used for three-dimensional modeling where more detailed stress analysis is required, such as in regions with complex stress gradients.

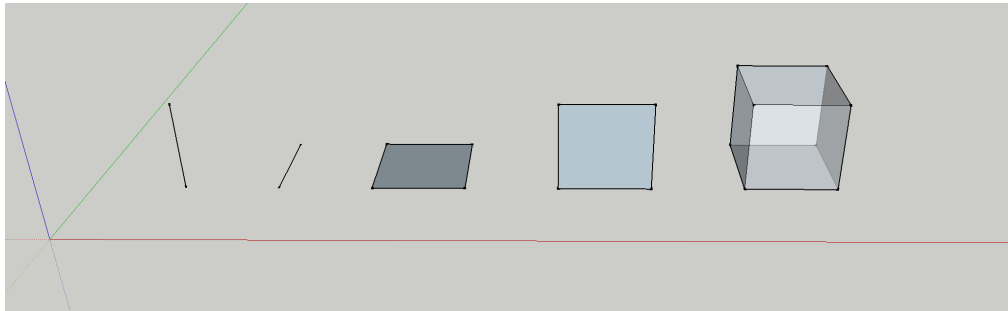


Figure 1: Line elements | Plate/shell elements | Solid (brick) element

In structural engineering, most problems can be modeled reasonably with one-dimensional line elements, or two-dimensional plate or shell elements. More complicated three-dimensional elements, such as solid (or brick) elements, have not been commonly used in most commercial software available for structural engineering today. When creating a model, these line and plate/shell elements with their associated properties are defined, as well as end nodes defined with translational or rotational degrees of freedom as shown in Figure 2.

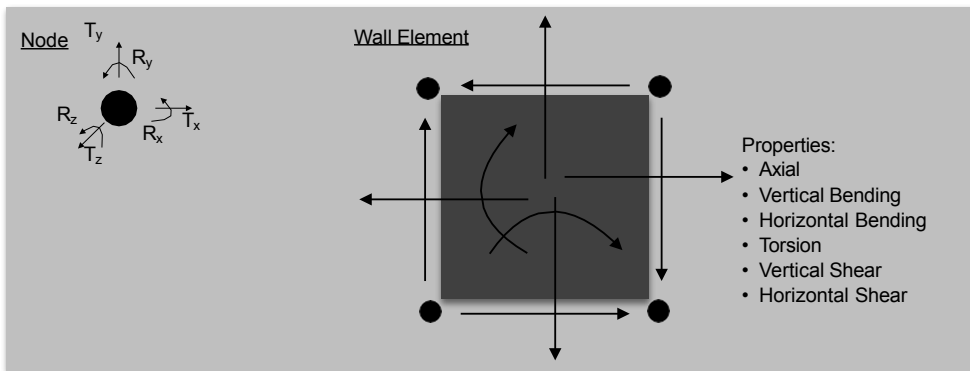


Figure 2: Node degrees of freedom and wall element properties

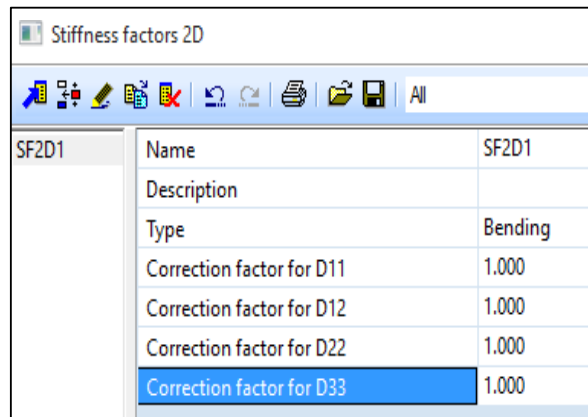
The properties designated to the line and plate elements must be defined to associate a reasonable stiffness with each element. Columns and beams (not masonry lintels) are able to be modeled with line elements, and walls and slabs are modeled with plate/shell elements. Many software programs allow the engineer to define the geometric boundaries of entire wall panels from movement joint to movement joint and discretize those large geometries into smaller finite elements by a process called meshing. Sometimes meshing is a manual process, and other times programs will offer automatic meshing as shown in Figure 3.



Figure 3: Wall Geometry with opening | Wall discretized into finite elements  
example of automatic meshing from RAM Elements

## Pre-Processing and Masonry Modeling

Many of the analyses used today assume thin plate theory for the plate elements along with linear elastic behavior for the elements. The elasticity of the material is described by a stress-strain curve, which shows the relation between internal force per unit area and the relative deformation. Linear elasticity is a simplification assuming linear relationships between the components of stress and strain which is valid only for stress states that do not produce yielding or fracture. Reinforced masonry and other reinforced concrete elements have the complication of not being elastic. Therefore, once a concrete element cracks, modeled steel reinforcement is then engaged in these elements. Of course, masonry is made up of different concrete components that closely mimic this behavior when it is reinforced. Many times, finite element software



Name	SF2D1
Description	
Type	Bending
Correction factor for D11	1.000
Correction factor for D12	1.000
Correction factor for D22	1.000
Correction factor for D33	1.000

Figure 4: Example of bending modification factors available in SCIA Engineer

gives us element modification factors to account for the reduced stiffness of the concrete or masonry element once it has cracked. In some programs this factor is automatically applied, in some it must be manually defined, and in others it is not an option. Some programs offer multiple element modification factors including bending in each direction, torsion, shear, and axial deformations as shown Figure 4 for SCIA engineer software. One must confirm that the element modification factor used to account for the reduced stiffness from cracking only applies to the bending stiffness in the direction of the cracked behavior and is not used with the shear stiffness or the axial stiffness of the element.

When the analysis program being used does not have an appropriate element modification factor, an adjustment to the actual properties of the element might be necessary. An adjustment may mean modifying the elastic modulus of the element. The elastic modulus is used to determine the stiffness for the element in each of the deformation categories, therefore an adjustment will impact the element in all properties of bending, shear, and axial deformation. This type of modification must be used with caution and may not always be appropriate.

Masonry is unique in that it is often reinforced in the vertical direction but left unreinforced in the horizontal direction. Therefore, the element may only span horizontally if it remains uncracked in that direction. If the anticipated design demand stresses are beyond the allowed cracking stresses, users should consider reducing stiffness by using a reduced element modification factor. Again, this emphasizes the need

for the modification factors to be considered separately in each orthogonal direction. When all of the factors are equal, the slab element behaves as an isotropic material, a material having the same properties in all directions. When the factors are different from each other, the slab elements behave as an orthotropic material, a material having different properties along its three perpendicular axes.

Care must be used when setting these stiffness factors. With certain combinations of factors, the structure can become unstable, and the results can become unreliable. Also, the interaction of the stiffness factors may be more complex than it appears upon first inspection.

Masonry design also requires custom material types within the software to account for attributes that are unique to this material, such as grouting only reinforced cells (partial grouting). This will affect how we must model masonry elements. Partial grouting affects both the loading aspect (from the self-weight contribution) of the finite element modeling as well as the stiffness of the masonry finite elements. There are some programs, such as RISA 3D, that account for partial grouting of the masonry wall, otherwise modifications must be made to the finite element properties (such as altering the actual thickness of the element). Again, there are pluses and minuses associated with modifying the thickness of an element to accommodate for the actual condition of partially grouted masonry. The axial and shear stiffness of the wall may be accurately modified, however the reduction to the bending stiffness of the finite elements would not be accurate and result in elements that are much weaker than they are in a real partially grouted wall. Therefore, engineering judgment must be used when the software does not account for partial grouting, and we are left to make modifications that may bring unintended consequences. It is also important to recognize that overall geometric wall modeling for masonry walls must account for the physical separation between walls due to control joints. RAM Elements allows for quickly separating linked wall panels (panels that share end nodes) into separate wall panels with unique end nodes. Whether there is a tool to create this separation, or the walls are manually modeled separately with unique end nodes, separation in the finite element model is required to ensure each wall is able to act independently from one another.

There are a few items to consider regarding finite element meshing. Finite element programs are based on plate elements that are quadrilateral (four nodes per plate/shell), and the ideal shape is a square. Without going into the finite element theory of why this is ideal, it is important to know that the further plate/shell elements are from a square, the less accurate the finite element approximations become. When considering the ideal size of the plate/shell elements when meshing (manually or automatically) a wall geometry, we need to consider the accuracy of the results, computational processing time, and the material being modeled. When considering accuracy, the finer the mesh (more smaller plates/shells), the higher the probability that the elements will be square, and the results will be more accurate. This is especially true in complex models. However, the smaller the mesh, the more plates/shells and nodes, the more time will be required for analysis. Even with the advances that have been made in software, finite element models with a very fine mesh can make computational time unreasonable. Lastly, considering the material properties, it could be argued that concrete and masonry have an inherent minimum element size due to what is referred to as the "chunkiness" of concrete. It is unreasonable to have differential movement between nodes that are closer together than the actual thickness of the masonry element. This is similar to evaluating one-way shear no closer than the depth of concrete element away from a support. Considering all of these size recommendations, there is also the point of diminishing returns. A model's approximate solution starts to converge on a solution, and using a finer mesh does not result in any changes to the overall solution. In general, the recommended maximum plate/shell size would be the span distance divided by ten and the minimum plate size should be no less than the thickness of the masonry wall. For example, a wall that is twelve inches thick and thirty feet tall would have a minimum plate size of twelve inches and a maximum

plate size of three feet (span/eight). Of course, there may be times when these guidelines must be reevaluated for unique situations, but in general they have been found to be a good starting point for determining plate/shell size in finite element models for walls.

As you can see, much care is involved when modeling masonry wall systems with finite element analysis programs to ensure all of the boundary conditions, stiffnesses of the elements, and weights of the elements are accurately accounted for in the development of the finite element model.

Some may wonder if all of this effort is worth it for a masonry wall. I would argue that it is definitely necessary if we want to understand the true behavior of complex wall systems, such as the in-plane shear wall capacity of perforated shear walls (wall panels with openings in the middle) and gain an even better understanding of the out-of-plane behavior in walls with openings.

Of course, modeling masonry finite elements is also essential in all of the following lateral analysis scenarios as well:

- Lateral dynamic analysis for any building with masonry lateral-resisting elements. Appropriate load and stiffness is required to understand the true dynamic behavior, which yields building fundamental periods.
- Lateral analysis load distribution (through rigid or semi-rigid diaphragms) between masonry and other systems or materials, such as concrete or structural steel frames

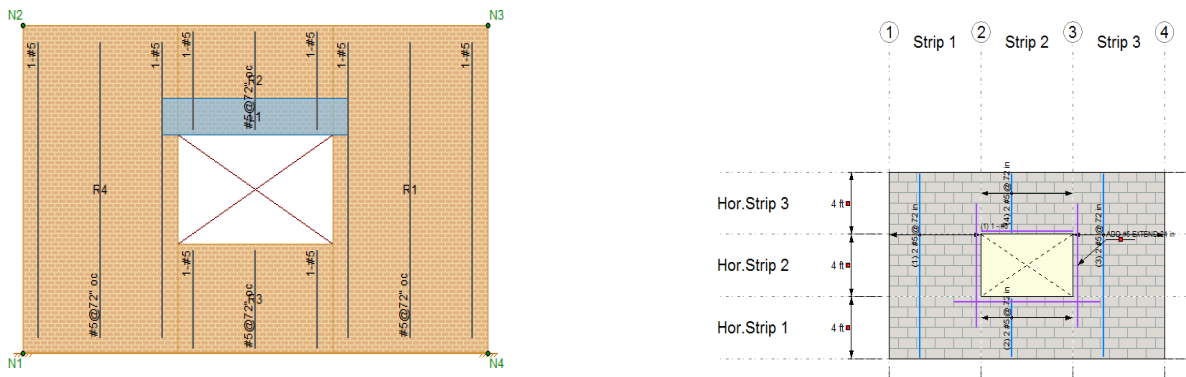


Figure 5: image above from RISA showing wall areas, image to the right of RAM Elements showing wall strips

Once the finite element model has been defined and the analysis is complete, then we need to turn our attention to the task of post-processing of finite element results and performing the design.

## Post-processing and Design

The next challenge involves taking the results from the finite element model and analysis and converting them into information that can be compared to code-defined maximum stresses or forces that determine the capacity of the masonry wall. Finite element programs for masonry combine the results of several plate/shell elements within geometric areas or strips of the model as defined by the user. Areas above openings are rationalized into an area that will be checked against lintel capacities. Engineers must study software programs and their combination (summation) of finite element results and make modifications when necessary. Figure 5 shows images of wall areas and strips from RISA and RAM Elements respectively.

Generally, structural engineering software will check for in-plane bending and shear capacity, out-of-plane bending and shear capacity, and axial capacity of masonry walls. Lintel shear and bending capacities will also be evaluated. Lintels (not in a finite element model) have traditionally been checked by assuming a simply supported "beam" element. Finite element approximation and design of the area above the openings are fundamentally different as the plates/shells in this area are interlocked by sharing nodes with the other surrounding elements of the wall. When evaluating bending moment in walls, software programs often evaluate only vertical bending and do not evaluate for horizontal bending and shear of masonry walls. Therefore, the user is left to manually check the horizontal bending moment against an unreinforced masonry bending capacity. If horizontal bond beams are used within a masonry wall, the horizontal bending moment may be manually checked against a reinforced bending capacity.

One may find some programs may not be able to correctly define the finite element model. If the program does not, the designer must decide if manual modifications can be made to the model without adversely affecting its other attributes and results. Further, evaluation of the post-processing design features of programs and design checks show that results are not always complete and must be supplemented with manual checks of the analysis results. Ultimately, careful evaluation when selecting software to be sure it is well-suited for your needs; however, additional calculations may be needed. Therefore, I would recommend thoroughly reviewing the element response to applied forces. The simplest and most revealing check can be made by animating the deflections of the elements. For example, a simply supported wall element should have a deflected animated shape that is a simple curve, and a wall with moments fixed at the top or with a parapet (cantilevered element above the roof) should have a compound curve. To review the forces in the element, a quick manual calculation should be within 20-25% of the anticipated forces in any particular element within a finite element model. Lastly, reviewing the reactions to the applied forces is a good study to make sure the elements are being modeled properly.

Of course, we also cannot forget about the fact that the finite element models we create as structural engineers often contain other materials and elements that are connecting to the masonry wall elements within our model. It is important to consider how those elements are connecting to masonry as shown in Figure 6. Consider items such as: are the beams (line elements) pinned at the end connecting into the masonry wall? Should the beam ends be offset from the centerline of the masonry wall panel? Are the shell/plate slab elements pinned or fixed to the masonry walls?

## Conclusion

Accurate finite element modeling of masonry demands attention to element discretization, material properties, boundary conditions, and analysis type. When implemented correctly, FEA provides deep insights into the structural behavior of masonry, enabling engineers to optimize designs for safety, efficiency, and cost-effectiveness. For complex projects involving lateral load resistance or dynamic response, FEA is indispensable for understanding and improving structural performance.

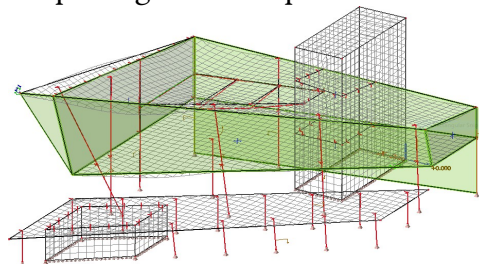


Figure 6: Example of complete finite element model with masonry walls, which has concrete slabs, steel beams connecting to the walls model from SCIA Engineer