Lateral Analysis: Right Way, Wrong Way with Software

Part 1 of the 2 part article

Introduction

The increasing ease of performing a Lateral Analysis of a structure is becoming a double-edged sword: there are many benefits which is an extremely good thing, but it can also be quite dangerous. I have seen my fair share of presentations and articles from seasoned engineers that warn about colleagues losing a sense of the real behavior of structures, or that engineers today simply do not know how to design structures without a computer. When I try to rationalize such comments, I quickly remember that the engineering curriculum remains similar to the education that engineers received 10, 20, or even 30 years ago. Many engineering programs focus on statics, dynamics, and material properties courses. So what explains this current perspective that today's engineers are not as grounded as their predecessors in their understanding of structural engineering design? I believe it is rooted in the thought that structural engineers are overly reliant on software programs and that software processes are replacing engineers' judgement.

Software programs should make us better engineers, not worse. They are tools and should be treated as such. Each program has unique and varying abilities to create a representation of the real structure, some with more features and options than others. However it is better to view all structural engineering software programs as incredible graphic user interfaces, able to solve complex sets of equations and run predefined formulae, rather than assume they are able to understand the complexity of the structures and the loads that need to be applied or be able to devise and create unique solutions. These tools provide us with the ability to solve problems very efficiently and iterate design options until we arrive at the best possible alternative, provided we accurately define the problem. I think that this - accurately defining the problem - is the main issue. Also, the isolated manner in which engineers work leads to few people thoroughly examining the design problem to ensure it is defined accurately. Individuals are then left to complete this work with no one watching, in an ever-increasing budget-constrained, schedule-cramped environment. I believe this is when the shortcomings of using software take root.

In the May 2016 issue of *Structure* magazine, I wrote an article on the use of Finite Element Method (FEM) for masonry, and perhaps future articles will feature the specific uses of other materials with the FEM. This article, however, is on the broader topic of lateral analysis, and the right way and wrong way to use FEM software programs. Therefore, this piece will not include material-specific recommendations;

 © 2024 FORSE Consulting, LLC

instead, it discusses lateral load generation, element (beam, column, wall) properties, diaphragm properties, loads on diaphragms, element connection to diaphragms, types of lateral analysis, and methods for quality assurance for the lateral analysis.

ints below the windows are normally even with the window jams because there is no need for an offset.

Brief Review of FEM Basics

FEM is the process of simplifying a real life structure, generally a continuum with infinite degrees of freedom, to a finite number of elements with unique material properties.

FEM is generally broken up into three steps:

1. Modeling: Pre-processing step where a user defines elements of the model, element connectivity, support conditions, and forces to represent various loading conditions.
2. Analysis: Processing step that requires little input from the user - generally users establish a few important parameters and then allow the software to solve vast sets of equations based on modeling.
3. Validation and Design: Post-processing, the step of interpreting and verifying the results of the analysis and then designing elements based on parameters determined by the material codes one uses.

Part 1 of this article will examine the first, and most important step for FEM, model generation.

Modeling

In defining a model, users establish one-dimensional (straight or curved) line elements with two end nodes, and/or two-dimensional plate (square, rectangular, or triangular planar shape) elements with nodes at each corner of the element. In the process of defining the elements and end nodes, some nodes will need to be defined as supports and others will remain as free nodes able to translate in three dimensional (X,Y, Z) degrees of freedom (DOF), and rotate about the three axes (RX, RY, RZ).

Support nodes generally have translational DOF fixed and or rotational DOF fixed. For the actual conditions the model represents, remember that most support conditions are less than the idealized fixed condition. In nearly all cases, it would be more accurate to specify the support as a force of resistance over a potential displaced distance; in other words, a spring support. This is true for both the translational and rotational degrees of freedom. Small differences in these support conditions may have a significant impact upon the lateral resistance of the assigned members. I am not saying that all foundations should be spring supports, but their true behavior should be considered, especially when a building has dissimilar lateral resisting elements such as moment frames and walls where lateral load distribution may be greatly affected by soil stiffness. However, when dealing with similar resisting elements, such as a building with equally sized and uniformly distributed braced frames, fixed supports or spring supports may have little effect on the outcome of the analysis. Here, a column's foundation can generally be modeled as a pinned support (free rotational DOF) as rotational stiffness may have less impact compared to vertical and horizontal translational stiffness.

Modeling - Line and Plate Elements

Section properties and member elasticity for line and plate elements need to be defined. Many of the software programs are based on a linear elastic analysis, which is sufficient for members that remain

 © 2024 FORSE Consulting, LLC

elastic under static loads such as dead load, live load, snow load, and even idealized wind loads on a model. However, additional material properties must be considered when an inelastic response is expected, such as for concrete elements that respond in a non-linear manner when the concrete cracks under tensile stress and engages reinforcement. Inelastic response is also expected for members designed to resist seismic dynamic loading with a response modification factor that is, in part, based on an inelastic response. This inelastic behavior of concrete is generally accounted for by reducing the stiffness with an element reduction factor. It is very important to realize that software tools do not make this modification automatically and require user input through several iterations of modeling and analysis (loads to the member change as stiffness changes and may require further member modification).

Many software programs allow the users to define the geometric boundaries of entire slab elements or wall panels 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. To a certain extent, the finer the mesh (smaller the elements) the better the finite element method can approximate the result. There is a point of diminishing returns in which finer meshes only result in a small percentage change in the results. It should be noted that finer meshes also produce significantly increased processing times for the finite element analysis. Having a mesh with nodes closer together than the thickness of the element (this is especially relevant with concrete materials) is generally unnecessary in most cases as it would be unreasonable to have differential movement between nodes spaced that close together.

With respect to the elements of the finite element model, the last piece of information to determine is connectivity. This can be simply defined as pinned (translational movement is shared between elements that share the same node) or fixed (translations and rotational degrees of freedom are maintained between the elements that share the node). Similar to the nodal degrees of freedom discussion earlier in the article, it is important to note that idealized connections between members are more accurately represented by a spring. Some deformation would occur between the two elements in a fixed joint, just as some rotational stiffness occurs for most connections specified as pinned. Just as with nodes, I am not suggesting all joints be connected with springs, but to consider the potential for each joint to act differently than the idealized condition and model accordingly. For example, I think it would be challenging to have a forty inch deep steel beam with thirteen rows of bolts act as a truly pinned end condition.

Another option for one-dimensional members in most software programs is the ability to shorten members based on the size of the members they are framing between. This allows users to create models to represent actual conditions and should be considered since all members have a physical size (width and depth). Users define members using centerline modeling, but then the program reduces member lengths and models rigid links to represent the fact that little to no deformation is likely to occur in this area. This changes the models in several ways. The vertical support members have reaction loading at face of the member creating eccentric loading, and the members that are supported are designed based on their actual length.

A few final words on elements - when we build finite element models, we can over-restrain members to nodes. For example, if there are two line finite elements that share a common node, they are connected and we can choose each members' connectivity at the node. If we release the rotational restraint for each member relative to global coordinate system, we receive a warning in finite element software, sometimes referred to as a local instability. The node needs to be fixed to one member or the other. But to avoid the error, I have seen engineers fix both axes (not one or the other), resulting in an over-connected model. Often at the end of a steel beam, only the strong axis moment is being designed as a moment connection.

© 2024 FORSE Consulting, LLC

The modeled weak axis moment fixity will play a part in the models' resistance to lateral loads. Although in most situations this may be relatively small, it can make a difference on load distribution. In certain models where large sections have a modeled weak axis moment conditions, it may "collect" relatively large loads that need to be addressed. Another element restraint that is often overlooked and unconventionally used for the purpose of eliminating instability warnings is torsional restraint (or rotational stiffness along the length of the member). This may not be an issue for sections such as concrete or closed steel section (HSS), but open steel sections do not resist torsion well. This is just another example of how a simple error in modeling may result in the collection of loads that is not being checked during design.

Modeling - Diaphragms

A very important criterion of lateral loading for buildings is the types of diaphragms that are defined. With nodes, line element columns, line element beams, and plate element walls, many programs offer the ability to define a diaphragm constraint instead of requiring plate element slabs to be modeled. Both rigid and non-rigid diaphragm types are idealized to simplify analysis. Rigid diaphragms fix the translation of all nodes of a similar elevation relative to one another; while non-rigid diaphragms allow free horizontal translation of one node to another. At this point, you might guess my hesitation to idealize modeling - is this necessary? In fact, we even name this approach of trying to capture the true diaphragm behavior as *semirigid diaphragm*. Geometric irregularities, lateral resisting elements with different materials and/or different types (walls and frames) or diaphragms with relatively large openings, should be defined as semirigid. Semirigid diaphragms complicate the stiffness of the finite element model by virtue of requiring many plate element slabs be defined, which again leads to increased time in analysis. Not every diaphragm needs to be semirigid. For example, diaphragms with similar and regular vertical lateral resistance elements and diaphragms with uniform and consistent slabs with few slab penetrations can likely be considered rigid.

When it comes to diaphragm action, caution must be used when there is a step in the diaphragm. Not only does the diaphragm action then require a vertical element (short wall, braces), but the diaphragm chords (generally at edges or extreme stress locations) must transition from one diaphragm level to another. Rigid diaphragms should not be specified as the same diaphragm on both levels of the step. The elements that transition the diaphragm forces must have the ability for the force to move through them to generate the true force transitions from one level of the step to the other.

A final thought about semirigid diaphragms is to also consider the element properties similar to other model elements. Settings for in-plane axial and shear stiffness, and out of plane shear and bending stiffness need to take into account either elastic or potentially inelastic behavior of semirigid diaphragm elements.

Modeling - Element Stiffness

All of these options for nodes, one and two-dimensional members, and diaphragms will change the lateral load distribution. The more strength and stiffness that is represented by an area of the model, the more the lateral load will be distributed to that area. It is important in the modeling phase to define actual

© 2024 FORSE Consulting, LLC

properties. Far too often, I see users taking shortcuts such as defining idealized support conditions, not defining section modification factors because it takes too much time, or defining material properties that are arbitrarily low in an effort to be conservative. Not only are each of these and other modeling shortcuts incorrect, but they will lead to inaccurate lateral load distribution which will result in some areas being assigned too much load (areas with too much stiffness) and other areas being assigned too little load, leading to unconservative designs (areas with too little stiffness). **There is no such thing as conservative modeling; all efforts should be made to be as accurate as reasonably possible when defining the model. The design step is the appropriate time for implementing conservative principles.**

I believe we all would like to think of ourselves as being progressive in our industry by using FEA software. I am suggesting much more progress be made. I think it would be wrong to blindly use software tools without fully understanding them, and also wrong if we do not fully utilize the tools. As my favorite author, C.S. Lewis, states, "We all want progress. But progress means getting nearer to the place where you want to be. And if you have taken a wrong turning, then to go forward does not get you any nearer. If you are on the wrong road, progress means doing an about-turn and walking back to the right road; in that case the man who turns back soonest is the most progressive man." If you find yourself on the wrong path, I ask that you reconsider your approach to modeling with FEA software. Part two of this article will discuss completing the modeling step by offering suggestions regarding applying loads to the model, comments on analyzing the model and when to review the results, and finally we will discuss designing members.

 © 2024 FORSE Consulting, LLC