

Jonathan Fraser
Senior Design
P07102

Introduction

Finite Element Analysis (FEA) is a computer simulation technique used in engineering analysis. It uses a numerical technique called the finite element method (FEM). In general, there are three phases in any computer-aided engineering task, which include pre-processing (defining the finite element model and environmental factors to be applied to it), analysis solver (solution of finite element model) and post-processing of the results.

The programs within the Mechanical Engineering Department that allow this style of analysis to be completed, are Wildfire 3.0, and ANSYS. For the purposes of assembly analysis, Wildfire 3.0 was used. This is also the program which all of these assemblies were modeled. The system within Wildfire 3.0, which allows for FEA to be accomplished is an integrated program called Mechanica. Mechanica is actually a suite of programs developed by PTC, however, RIT only has the structural and thermal portions of that suite. The structural simulation can complete linear static stress, modal, buckling, and large deformation analysis.

FEA: Element Size and Definition Differences

Each FEA system offered within the department uses a system of divided regions within each assembly or part; these regions are the finite elements which compose the mesh that is analyzed in later steps. Generally, lines intersect to create nodes; the elements produced from said lines are either triangles or quadrilaterals in 2-D and are tetrahedral or brick like elements in 3-D structures. In each case, the partial differential equation's output dependent variable is computed at the nodes, therefore the solution is only given at those places. One way around this is to create a large number of nodes similar to ANSYS; however, Mechanica uses a method called interpolating polynomials, this allows for the analysis time to be greatly reduced without needing to refine the mesh after each pass. In both systems, ANSYS and Mechanica, a similar set of conditions represented by several linear algebraic equations must be satisfied.

The majority of FEA programs will combine all of the individual elements into a mesh and then convert the problem from a set of continuous differential equations into a large set of simultaneous linear algebraic equations. This system will have several thousand equations in it, and the solution of the simultaneous linear algebraic equations represents an approximation of the continuous differential equations, which represented the mesh. This approximation can call into question the accuracy of the results. In theory, a mesh of infinite number will create the best approximation, however this would produce a system of simultaneous linear algebraic equations of infinite size as well.

The actual steps the programs take in solving a system of simultaneous linear algebraic equations can be complex, however they can be reduced to the following. The dependent variable in the governing partial differential equation is the displacement from a reference, which is usually the unloaded position. The material strain, displacement per

unit length, is then computed from the displacement by taking the derivative with respect to its initial position. Finally, the stress components at any point in the material are computed from the strain at that point; thus, if the interpolating polynomial for the spatial variation of the displacement field is linear within that small element, then the strain and stress will be constant within that element since the derivative of the linear function representing that element is also constant.

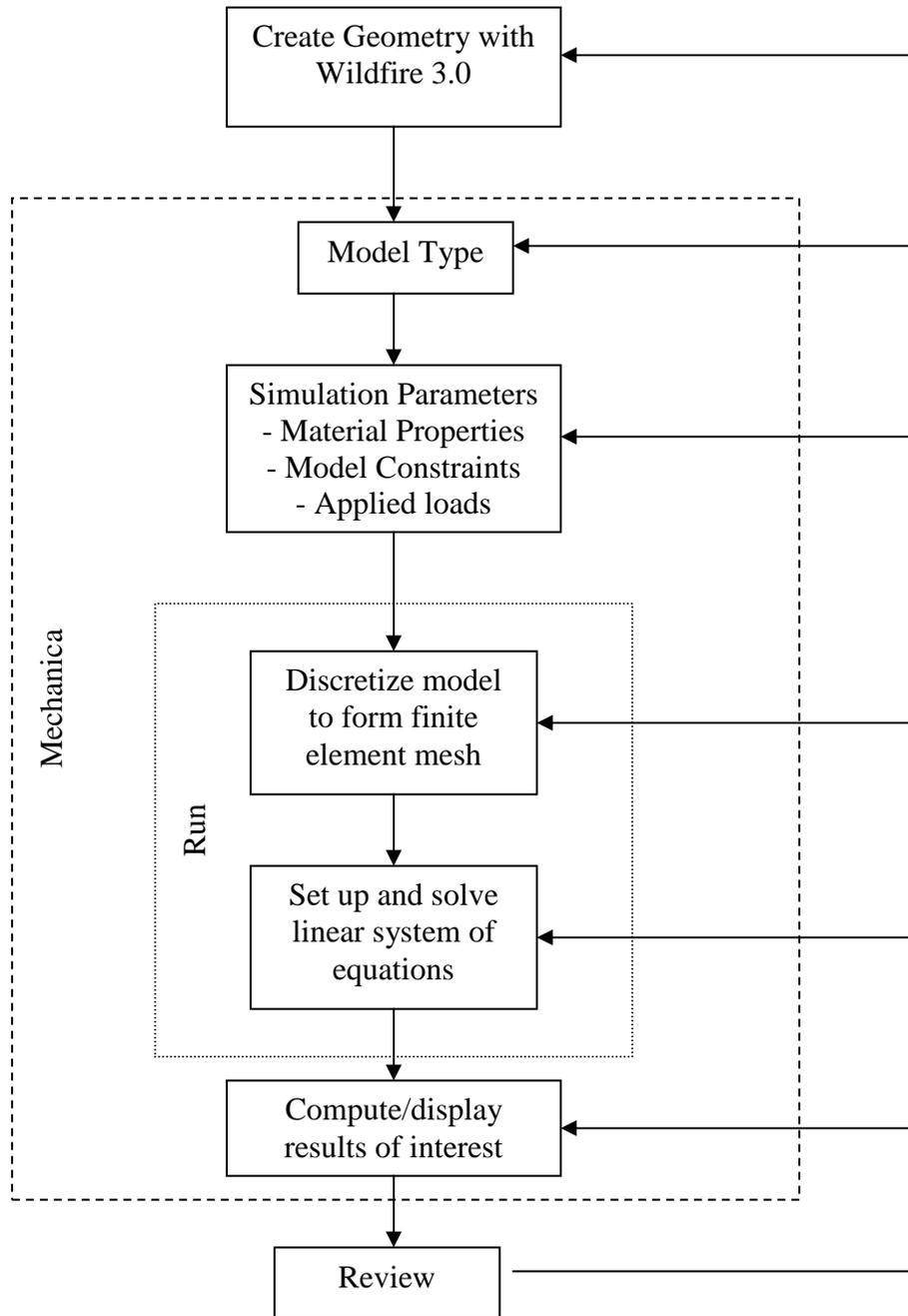
Generalized Processing Steps

Often times in order to obtain results in a finite amount of time; several assumptions or simplifications must be made in order to have a final workable model. Workable means the FEA model must allow for the computation of the results of interest with sufficient accuracy and with acceptable time and resource usage. The last two have become important considerations because the computers that these analyses are run on are both in open labs and a predetermined amount of resources (RAM, hard drive space, and processing power) dedicated to each.

In simplifying the model the user must make several assumptions. One such assumption is that the materials assigned to the model are homogeneous, isotropic, and free of internal defects or flaws, another is to ignore aspects of the geometry that pose no anticipated effect on the results, such as chamfered edges along the outside of the part. Ignoring cosmetic features and assuming uniform materials are common within industry are often the first set of assumptions used to simplify a model.

After making those assumptions, the model can be called a simplified physical model. The next step would be making further assumptions to create a mathematical model. This set of assumptions includes linear material properties and idealized loading conditions. To idealize the loading conditions, the loading must be steady, and placed on perfectly fixed points. At this time the model is converted into one or more differential equations that describe the variation in characteristics within the boundaries of the model.

The next step is creating an FEA model from the mathematical model. This is where the conversion from one or more differential equations into an equal number of simultaneous linear algebraic equations occurs. A run processor actually performs the solution to the proposed problem, this run processor is an FEA engine that will use special numerical techniques and algorithms to exploit various properties within the system of equations. The output can be displayed graphically showing displaced shape, stress distribution, mode shapes and many other important features.



Element type

Convergence of H-elements

The element type used is the key difference between Mechanica and ANSYS. ANSYS uses a classical approach in the convergence of what is called an H-element. The H-element is a low order interpolating polynomial, which causes significant ramifications. The stress analysis is the primary solution for the variables at the nodes. The interpolating functions with this element type are linear within each element. Strain is obtained by taking the derivatives of the displacement field and the stress computed from the material strain. For a first order interpolating polynomial within each element, this would cause the strain and stress components to be constant everywhere, which cause a discontinuity in the stress field between elements, and will lead to inaccurate and unrealistic values for the maximum local and global stresses. The use of low order elements lead to the greatest inaccuracies at the areas of greatest interest, which are the areas where large stress gradients would occur in a real object.

This is why ANSYS requires the refinement of mesh around these areas of interest. The process of mesh refinement is called convergence analysis, on H-elements; this style of convergence is called H-convergence. Often times, this convergence analysis leads to a larger and larger set of differential equations, and then simultaneous linear algebraic equations. Another drawback of the H-element is its inability to adapt to shape extremes in terms of skew, rapid size variation, and large aspect ratio.

Convergence of P-elements

Mechanica uses what is called the P-element. The P-element is unique because convergence is obtained by increasing the order of the interpolating polynomials in each individual element. The mesh will stay the same for every pass. The FEA engine can recognize the areas where high gradients occur and those elements have their order of the interpolating polynomials increased. This allows for the monitoring of expected error in the solution and then automatically increases polynomial order as needed. The accepted level of error in a solution using the highest settings is less than ten percent; often times, an analysis with less than two percent error can be accomplished. Also, the mesh restrictions for this element size are not nearly as stringent for the P-element.

Conclusion

The use of the built in FEA solver incorporated into Wildfire 3.0 is an effective alternative to ANSYS. The academic version of ANSYS does not allow for the importation of large assemblies, thus it would be required that we rebuild each model after each design iteration, therefore Wildfire 3.0 was used. While the engineering industry uses ANSYS as the standard FEA system, the improvements in P-elements and processing engines have made the solver within Wildfire 3.0, Mechanica a viable system for providing excellent finite element analysis.