tetrahedral elements. Hexahedral elements are generally more accurate, but can be more challenging to mesh automatically. Tetrahedral elements can match hexahedral element performance by using more (smaller) elements, and tetrahedral elements are much easier to mesh automatically. SW Simulation uses only tetrahedral elements for solid studies. An example of the small two-dimensional geometric boundary error due to different curved shapes is seen in Figure 1-3 where a arc and a parabola pass through the same three points.

Figure 1-3 Linear or parabolic elements never match circular shapes

1.5 Stages of Analysis and Their Uncertainties

An FEA always involves a number of uncertainties that impact the accuracy or reliability of each stage of an FEA and its results. The book, *Building Better Products with Finite Element Analysis* by Adams and Askenazi [1] gives an outstanding detailed description of most of the real-world uncertainties associated with solid mechanics FEA. All engineers conducting stress studies should read it. That book also points out how poor solid modeling skills can adversely affect the ability to construct meshes for any type of FEA. Here, the most important FEA uncertainties are highlighted.

The typical stages of an FEA study are listed below:

1. Construct the part(s) in a solid modeler. It is surprisingly easy to accidentally build flawed models with tiny lines, tiny surfaces or tiny interior voids. The part will look fine, except with extreme zooms, but it may fail to mesh. Most systems have checking routines that can find and repair such problems before you move on to an FEA study. Sometimes you may have to export a part, and then import it back with a new name because imported parts are usually subjected to more time consuming checks than "native" parts. When multiple parts form an assembly, always mesh and study the individual parts before studying the assembly. Try to plan ahead and introduce split lines into the part to aid in mating assemblies and to

locate load regions and restraint (or fixture or support) regions. Today, construction of a part is probably the most reliable stage of any study.

2. Defeature the solid part model for meshing. The solid part may contain features, like a raised logo, that are not necessary to manufacture the part, or required for an accurate analysis study. They can be omitted from the solid used in the analysis study. That is a relative easy operation supported by most solid modelers (such as the "suppress" option in SW) to help make smaller and faster meshes. However, it has the potential for introducing serious, if not fatal, errors in a following engineering study. This is a reliable modeling process, but its application requires engineering judgment. For example, removing small radius interior fillets can greatly reduces the number of elements and simplifies the mesh generation. But, that creates sharp reentrant corners that can yield false infinite stresses. Those false high stress regions may cause you to overlook other areas of true high stress levels. Small holes lead to many small elements (and long run times). They also cause stress concentrations that raise the local stress levels by a factor of three or more. The decision to defeature them depends on where they are located in the part. If they lie in a high stress region you must keep them. But defeaturing them is allowed if you know they occur in a low stress region. Such decisions are complicated because most parts have multiple possible loading conditions and a low stress region for one load case may become a high stress region for another load case.

3. Combine multiple parts into an assembly. Again, this is well automated and reliable from the geometric point of view and assemblies "look" as expected. However, geometric mating of part interfaces is very different for defining their physical (displacement, or temperature) mating. The physical mating choices are often unclear and the engineer may have to make a range of assumptions, study each, and determine the worst case result. Having to use physical contacts makes the linear problem require iterative solutions that take a long time to run and might fail to converge.

4. Select the element type. Some FEA systems have a huge number of available element types (with underlying theoretical restrictions). The SolidWorks system has only the fundamental types of elements. Namely, truss elements (bars), frame elements (beams), thin shells (or flat plates), thick shells, and solids. The SW simulation system selects the element type

(beginning in 2009) based on the shape of the part. The user is allowed to covert a non-solid element region to a solid element region, and visa versa. Knowing which class of element will give a more accurate or faster solution requires training in finite element theory. At times a second element type study is used to help validate a study based on a different element type.

5. Mesh the part(s) or assembly, remembering that the mesh solid may not be the same as the part solid. A general rule in an FEA is that your computer never has enough speed or memory. Sooner or later you will find a study that you cannot execute. Often that means you must utilize a crude mesh (or at least crude in some region) and/or invoke the use of symmetry or anti-symmetry conditions. Local solution errors in a study are proportional to the product of the local element size and the gradient of the secondary variables (i.e., gradient of stress or heat flux). Therefore, you exercise mesh control to place small elements where your engineering judgment estimates high stress (or flux) regions, as well as large elements in low stress regions. The local solution error also depends on the relative sizes of adjacent elements. You do not want skinny elements adjacent to big ones. Thus, automatic mesh generators have options to gradually vary adjacent element

restraint (fixture) regions formed by split lines, even if such splits are not needed for manufacturing the parts. The mesh typically should have refinements at source or load regions and support regions.

A mesh must look like the part, but that is not sufficient for a correct study. A single layer of elements filling a part region is almost never enough. If the region is curved, or subjected to bending, you want at least three layers of quadratic elements, but five is a desirable lower limit. For linear elements you at least double those numbers.

Most engineers do not have access to the source code of their automatic mesh generator. When the mesher fails you frequently do not know why it failed or what to do about it. Often you have to re-try the mesh generation with very large element sizes in hopes of getting some mesh results that can give hints as to why other attempts failed. The meshing of assemblies often fails. Usually the mesher runs out of memory because one or more parts had a very small, often unseen, feature that causes a huge number of tiny elements to be created (see the end of this chapter). You should always attempt to mesh each individual part to spot such problems before you attempt to mesh them as a member of an assembly.

Automatic meshing, with mesh controls, is usually simple and fast today. However, it is only as reliable as the modified part or assembly supplied to it. Distorted elements usually do not develop in automatic mesh generators, due to empirical rules for avoiding them. However, distorted elements locations can usually be plotted. If they are in regions of low gradients you can usually accept them.

You should also note that studies involving natural frequencies are influenced most by the distribution of the mass of the part. Thus, they can still give accurate results with meshes that are much cruder than those that would be acceptable for stress or thermal studies.

6. Assign a linear material to each part. Modern FEA systems have a material library containing the "linear" mechanical, thermal, and/or fluid properties of most standardized materials. They also allow the user to define custom properties. The property values in such tables are often misinterpreted to be more accurate and reliable than they actually are. The reported property values are accepted average values taken from many tests. Rarely are there any data about the distribution of test results, or what standard deviation was associated with the tests. Most tests yield results that follow a "bell shaped" curve distribution, or a similar skewed curve. The tests for stainless steel tend to have narrow distributions, like that on the left in Figure 1-4, while the results for cast iron have wider distributions.

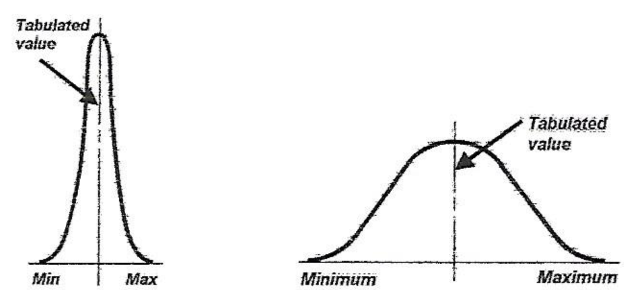

Figure 1-4 Typical distributions of properties of steel (left) and cast iron

When you accept a tabulated property value as a single number to be used in the FEA calculation remember it actually has a probability distribution associated with it. You need to assign a contribution to the total factor of safety to allow for variations from the tabulated property value.

The values of properties found in a material table can appear more or less accurate depending on the units selected. That is an illusion often caused by converting one set of units to another, but not truncating the result to the same number of significant figures available in the actual test units. For example, the elastic modulus of one steel is tabulated from the original test as 210 MPa, but when displayed in other units it shows as 30,457,924.92 psi. Which one do you believe to be the experimental accurate value; the 3 digit value or the 10 digit one? The answer affects how you should view and report stress results. The axial stress in a bar is equal to the elastic modulus times the strain, $\sigma = E\,\varepsilon$. Thus, if E is only known to three or four significant figures then the reported stress result should have no more significant figures.

Material data are usually more reliable than the loading values (considered next), but less accurate that the model or mesh geometries.

7. Select regions of the part(s) to be loaded and assign load levels and load types to each region. In mathematical terminology, load or flux conditions on a boundary region are called Neumann boundary conditions, or non-essential conditions. The geometric regions can be points (in theory), lines, surfaces, or volumes. If they are not existing features of the part, then you should apply split lines to the part to create them before activating the mesh generator. Point forces, or heat sources, are common in undergraduate studies, but in an FEA they cause false infinite stresses, or heat flux. If you include them do not be mislead by the high local values. Refining the mesh does not help since the smallest element still reports near infinite values.

   In reality, point loads are better modeled as a total force, or pressure, acting over a small area formed by prior split lines. Saint Venant's Principle states that two different, but statically equivalent, force systems acting on a small portion of the surface of a body produce the same stress distributions at distance large in comparison with the linear dimensions of the portion where the forces act. That also implies that concentrated sources quickly become re-distributed, as seen in Figure 1-5. There a single axial forced on the right end has been replaced by a small region of constant pressure. The other end stresses are essentially zero. Within the distance of about one depth the axial stress has re-distributed to an essentially constant value in the remainder of the part.

Figure 1-5 St. Venant's principle: local effects quickly die out

In undergraduate statics and dynamics courses engineers are taught to think in terms of point forces and couples. Solid elements do not accept pure couples as loads, but statically equivalent pressures can be applied to solids and yield the correct stresses. Indeed, a couple at a point is almost impossible to create, so the distribution of pressures is probably more like the true situation.

The magnitudes of applied loads are often guesses, or specified by a governing design standard. For example, consider a wind load. A building standard may quote a pressure to be applied for a given wind speed. But, how well do you know the wind speed that might actually be exerted on the structure? Again, there probably is some type of "bell curve" around the expected average speed. You need to assign a contribution to the total factor of safety to allow for variations in the uncertainty of the load value or actual spatial distribution of applied loads.

Loading data are usually less accurate than the material data, but much more accurate that the restraint or supporting conditions considered next.

8. Determine (or more likely assume) how the model interacts with the surroundings not included in your model. These are the restraint (support, or fixture) regions. In mathematical terminology, these are called the essential boundary conditions, or Dirichlet boundary conditions. You cannot afford to model everything interacting with a part. For many decades engineers have developed simplified concepts to approximate surroundings adjacent to a model to simplify hand calculations. They

include roller supports, smooth pins, cantilevered (encastre, or fixed) supports, straight cable attachments, etc. Those concepts are often carried over to FEA approaches and can over simplify the true support nature and lead to very large errors in the results.

The choice of restraints (fixations, supports) for a model is surprisingly difficult and is often the least reliable decision made by the engineer. Small changes in the supports can cause large changes in the results. It is wise to try to investigate a number of likely or possible support conditions in different studies. When in doubt, try to include more of the surrounding support material and apply assumed support conditions to those portions at a greater distance from critical part features.

You need to assign a contribution to the total factor of safety to allow for variations in the uncertainty of how or where the actual support conditions occur. That is especially true for buckling studies.

9. Solve the linear system of equations, or the eigenvalue problem. With today's numerical algorithms the solution of the algebraic system or eigen-system is usually quite reliable. It is possible to cause ill-conditioned systems (large condition number) with meshes having bad aspect ratios, or large elements adjacent to small ones, but that is unlikely to happen with automatic mesh generators.

10. Check the results. Are the reactions at the supports equal and opposite to the sources you thought that you applied? Are the results consistent with the assumed linear behavior? The engineering definition of a problem with large displacements is one where the maximum displacement is more than half the smallest geometric thickness of the part. The internal definition is a displacement field that significantly changes the volume of an element. That implies the element geometric shape noticeably changed from the starting shape, and that the shape needs to be updated in a series of much smaller shape changes. Are the displacements big enough to require re-solution with large displacement iterations turned on? Have you validated the results with an analytic approximation, or different type of finite element? Engineering judgments are required.

11. Post-process the solution for secondary variables. For structural studies you generally wish to document the deflections, reactions and stresses. For

thermal studies you display the temperatures, heat flux vectors and reaction heat flows. With natural frequency models you show (or animate) a few mode shapes. In graphical displays, you can control the number of contours employed, as well as their maximum and minimum ranges. The latter is important if you want to compare two designs on a single page. Limit the number of digits shown on the contour scale to be consistent with the material modulus (or conductivity, etc.). Color contour plots often do not reproduce well, but graphs do, so learn to use them in you documentation.

12. Determine (or more likely assume) what failure criterion applies to your study. This stage involves assumptions about how a structural material might fail. There are a number of theories. Most are based on stress values or distortional energy levels, but a few depend on strain values. If you know that one has been accepted for your selected material then use that one (as a contribution to the overall factor of safety). Otherwise, you should evaluate more than one theory and see which is the worst case. Also keep in mind that loading or support uncertainties can lead to a range of stress levels, and variations in material properties affect the strength and unexpected failures can occur if those types of distributions happen to intersect, as sketched in Figure 1-6.

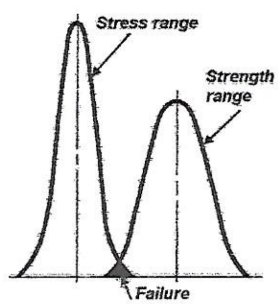

Figure 1-6 Overlap of stresses and material strengths can cause failure

14. Optionally, post-process the secondary variables to measure the theoretical error in the study, and adaptively correct the solution. This converges to an accurate solution to the problem input, but perhaps not to the problem to be solved. Accurate garbage is still garbage.

15. Document, report, and file the study. The part shape, mesh, and results should be reported in image form. Assumptions on which the study was

based should be clearly stated, and hopefully confirmed. The documentation should contain an independent validation calculation, or two, from an analytical approximation or an FEA based on a totally different element type. Try to address the relative uncertainties of the main analysis stages, as summarized in Figure 1-7.

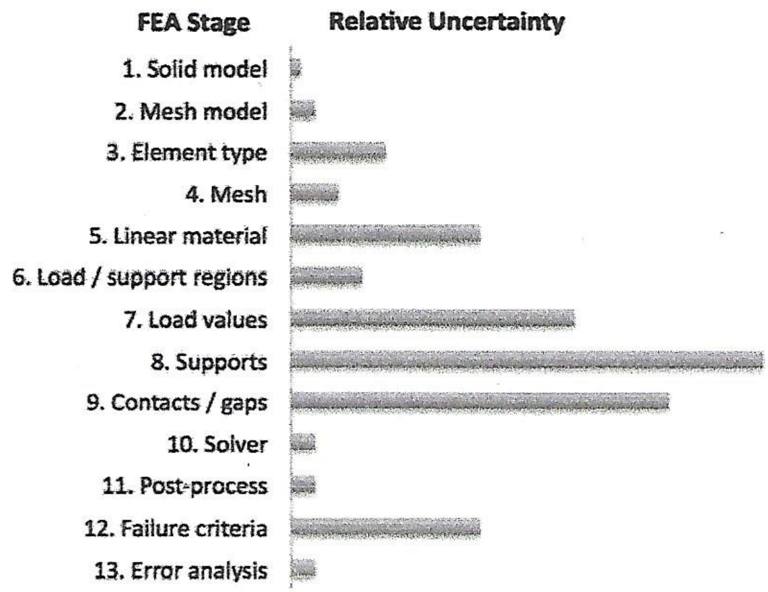

Figure 1-7 Relative uncertainty of major modeling stages

Technical communication and documentation is always important. In America, engineers are supposed to retain their calculations for at least seven years. Will your report be clear and helpful if you have to defend it years later? Paper hardcopies are the most reliable for long term storage. (Can you read the electronic media you used five years ago?)

You usually assume that the materials are linear. If not (creeping, hyperelastic, inelastic, plastic, viscoelastic, etc.), define the appropriate material data and the nonlinear equations to be solved. Then the matrix system becomes non-linear. Your original results check may lead you to conclude that the problem is actually an iterative one due to large displacements, or the need to insert physical contact interfaces.