

Beginning Simulation Studies

In this course the simulation (finite element analysis) capabilities of SolidWorks for stress analysis, dynamic response, and vibrations will be utilized as 'black box' tools without first mastering the theory of finite element analysis. However, there are a few important topics that you should know. For stress or displacement analysis they include:

1. Knowing the differences between brittle and ductile materials, and the failure criterion that should be checked for either material type. Brittle materials, like some cast iron, fail suddenly at small strains while ductile materials, like some steels, fail gradually often with large strains and visible distortions. Brittle materials will not absorb much energy before failing. By contrast, ductile materials absorb large amounts of energy before failure, mainly through plastic strains that caused permanent distortions of the part.

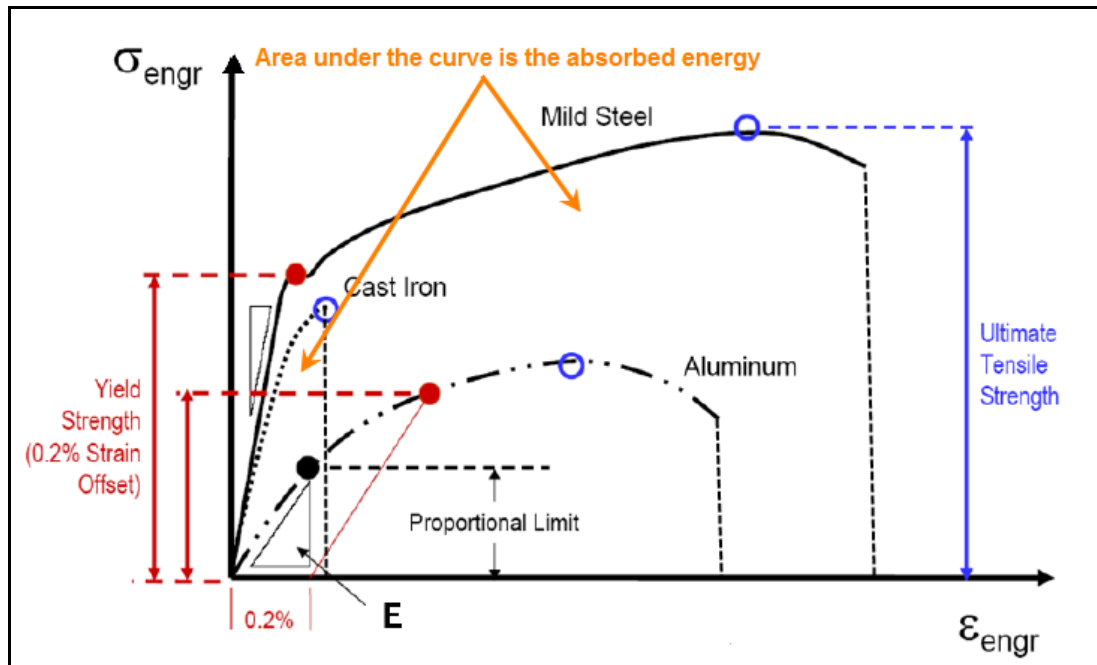


Figure 1 Typical stress-strain curves for brittle and ductile materials

2. Which of the available 17 stress components should you select for plotting and discussion in your report? The selected stress must be clearly named. There are 6 Cartesian stress components at a point which can alternately be expressed as 6 cylindrical coordinates stress components. Either set of those 6 stresses yields the same additional 3 Principal Stresses. Those 3 Principal Stresses then yield definitions of the Stress Intensity, Maximum Shear Stress, Maximum Normal Stress, Mohr-Coulomb Stress, and the von Mises Stress (distortional energy measure), and others.

The default system template always supplies a plot of the scalar von Mises Stress, which is commonly checked for ductile materials. However, that does not mean that the distortional energy failure criterion should be checked for your material or that it is the only value to display in any given application. That criterion is often called a stress since it has the units of stress, but it is actually a measure of how much energy is stored while distorting the material. The von Mises ductile failure criteria is a positive scalar (with the units of stress) obtained from the square root of the sum of the squares of the differences in the three principal stresses.

$$\sigma_{vM} = \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2}$$

$$\sigma_{vM} = \sqrt{(SX - SY)^2 + (SY - SZ)^2 + (SX - SZ)^2 + 6(TXY^2 + TXZ^2 + TYZ^2)}$$

When the von Mises value equals the yield stress in tension then a ductile material is deemed to fail.

When you 'Define a Stress Plot', from the list below, you should know why that stress is important (at least in some region) and explain in your report why it was evaluated and what you concluded from it. In undergraduate courses we tend to consider the classes of axial stress, bending stress, torsional stress, shear stress and bearing stress. A FEA computes the three-dimensional stress at a point. If you believe that the magnitude is due to one of the above classes then that may govern your choice of stress components.

For example, if a region is long and skinny like a beam then normal stress at the surface in the direction of the length may govern. Likewise, the maximum shear stress is at the centerline of such a member and it can govern for some materials. Thus, you should check both stress components in that region.

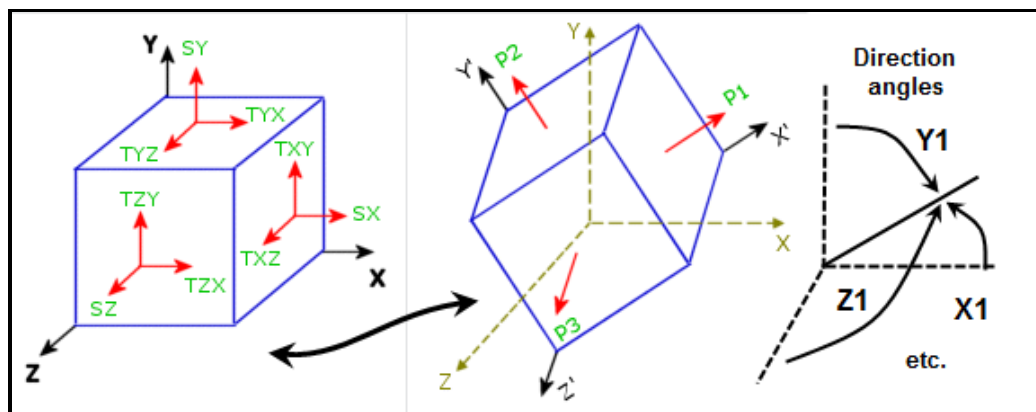
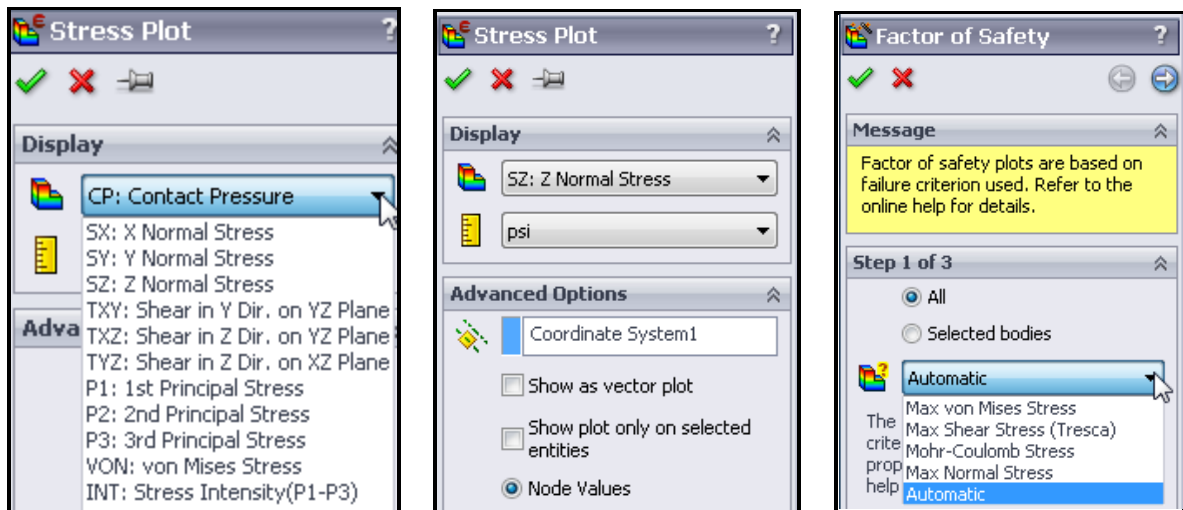


Figure 2 Options for selecting stress components or material failure laws

3. How do you amplify your static loads to account for impact loads or cyclic (fatigue) loads in a static stress analysis? The posted impact analysis help file shows that a vertical Impact Factor ranging typically from 2 to 1,500 must multiply an applied load to define the equivalent static force. Otherwise, it is necessary to run a time-history dynamic study to obtain the maximum displacements and the maximum stresses in a part. For horizontal impacts of a mass the horizontal Impact Factor can be less than unity and define an equivalent static horizontal force less than the full weight of the impacting mass. There is also a TK Solver formulation that can solve many common body stiffnesses and is easily extended to others.
4. How to control your mesh? Generally the local error in a solution at a point is proportional to the product of the size of the element and the gradient of the governing stress component. Therefore, where you see rapidly changing stress values use Mesh Control to produce smaller elements there. That may require that in the solid model you add 'split lines' to create regions to receive smaller elements. In addition to the element size, that feature also lets you control how fast the neighboring element grow in size (the default is a factor of 1.5). To make the element sizes grow slowly the use a 1.1 or 1.2 factor instead.

Any sudden change in shape will locally increase the stresses and heat flow in that region. That is called Stress Concentration in stress analysis. Analytical solutions have been developed to define the "stress concentration factor, k " that multiplies a uniform local stress, σ_L , to define the maximum stress at the change in geometry:

$$\sigma_{max} = k\sigma_L$$

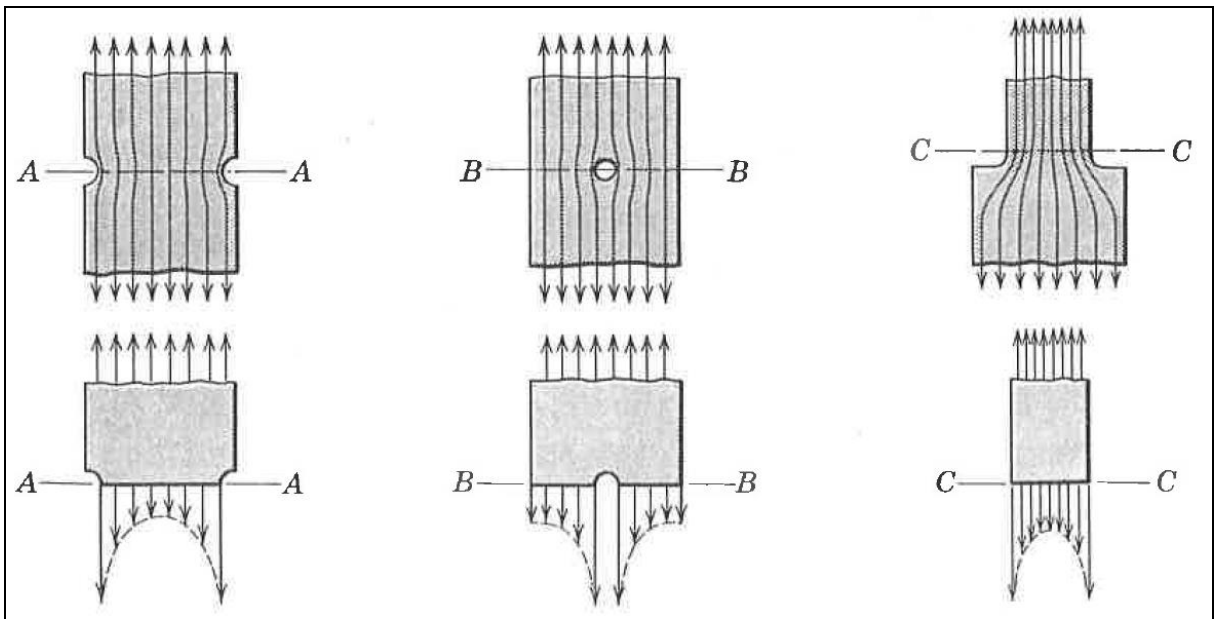


Figure 3 Sudden geometry changes concentrate (increase) local stress components

For example, a small circular hole in a wide plate increases the normal stress at the edge of the hole by a factor of 3. Some of the analytic values of stress concentration for common geometries are given in Figure 4.

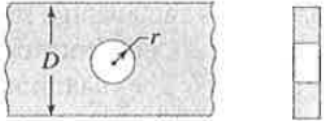
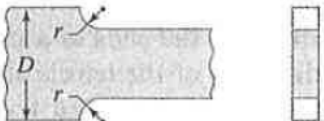
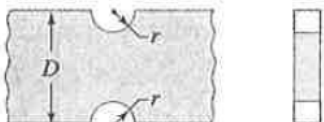


<p>(a) Circular hole in rectangular bar Axial tension $k = 3.00 - 3.13 \left(\frac{2r}{D}\right) + 3.66 \left(\frac{2r}{D}\right)^2 - 1.53 \left(\frac{2r}{D}\right)^3$ In-plane bending $k = 2$ (independent of r/D)</p>	
<p>(b) Shoulder with circular fillet in rectangular bar Axial tension $k = 1.976 - 0.385 \left(\frac{2r}{D}\right) - 1.022 \left(\frac{2r}{D}\right)^2 + 0.431 \left(\frac{2r}{D}\right)^3$ In-plane bending $k = 1.976 - 1.925 \left(\frac{2r}{D}\right) + 0.906 \left(\frac{2r}{D}\right)^2 + 0.0430 \left(\frac{2r}{D}\right)^3$</p>	
<p>(c) Semicircular notches in rectangular bar Axial tension $k = 3.065 - 3.370 \left(\frac{2r}{D}\right) + 0.647 \left(\frac{2r}{D}\right)^2 + 0.658 \left(\frac{2r}{D}\right)^3$ In-plane bending $k = 3.065 - 6.269 \left(\frac{2r}{D}\right) + 7.015 \left(\frac{2r}{D}\right)^2 - 2.812 \left(\frac{2r}{D}\right)^3$</p>	
<p>(d) Shoulder with circular fillet in circular shaft Axial tension $k = 1.990 - 2.070 \left(\frac{2r}{D}\right) + 1.938 \left(\frac{2r}{D}\right)^2 - 0.857 \left(\frac{2r}{D}\right)^3$ Bending $k = 1.990 - 2.429 \left(\frac{2r}{D}\right) + 2.057 \left(\frac{2r}{D}\right)^2 - 0.619 \left(\frac{2r}{D}\right)^3$ Torsion $k = 1.580 - 1.796 \left(\frac{2r}{D}\right) + 2.000 \left(\frac{2r}{D}\right)^2 - 0.784 \left(\frac{2r}{D}\right)^3$</p>	
<p>(e) Semicircular groove in circular shaft Axial tension $k = 3.04 - 5.42 \left(\frac{2r}{D}\right) + 6.27 \left(\frac{2r}{D}\right)^2 - 2.89 \left(\frac{2r}{D}\right)^3$ Bending $k = 3.04 - 7.236 \left(\frac{2r}{D}\right) + 9.375 \left(\frac{2r}{D}\right)^2 - 4.179 \left(\frac{2r}{D}\right)^3$ Torsion $k = 2.000 - 3.394 \left(\frac{2r}{D}\right) + 4.231 \left(\frac{2r}{D}\right)^2 - 1.837 \left(\frac{2r}{D}\right)^3$</p>	

Figure 4 Typical analytic stress concentration values

Also, regions subject to bending should have at least three solid elements through the local thickness. The program does have automatic h-adaptive solvers which can iteratively change the mesh to reach a specified level of accuracy in the energy norm.

- How much should you reduce the allowable governing stress to account for knowing that your loadings will be applied tens of thousands of cycles to tens of millions of cycles? To do that you need specific material testing data in the form of a governing stress vs number of cycles plot. That plot is called a "S-N Curve". SolidWorks includes such plot data for a very few of its tabulated material properties. See the help file on cyclic loading for empirical relations for setting an allowable stress level from the number of loading cycles expected over the life of a design. Once a valid static analysis has been obtained, SolidWorks has a module that will accept that analysis along with a definition of the periodic nature of the forces and compute the estimated number of loading cycles that will cause failure. Typical S-N data for common materials are shown below.

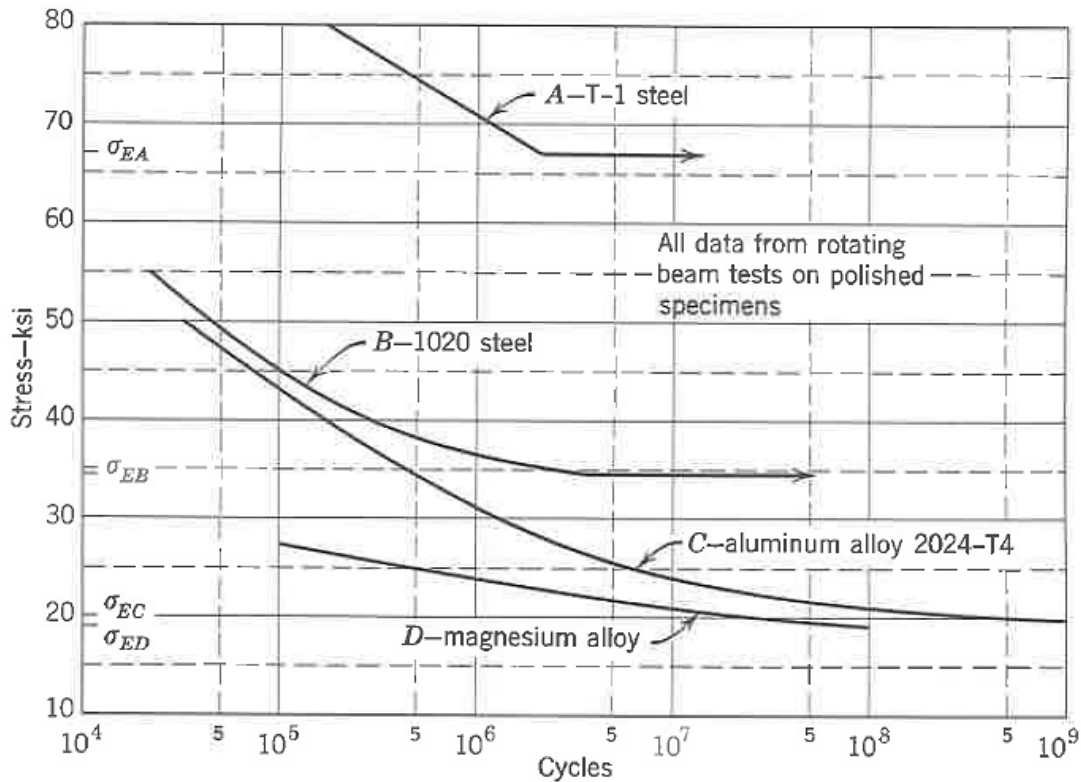


Figure 5 Typical limit stress versus cycles of loading

For a single material, the S-N data tend to be grouped as a band as seen below for a typical steel (and normalized with respect to the tensile strength):

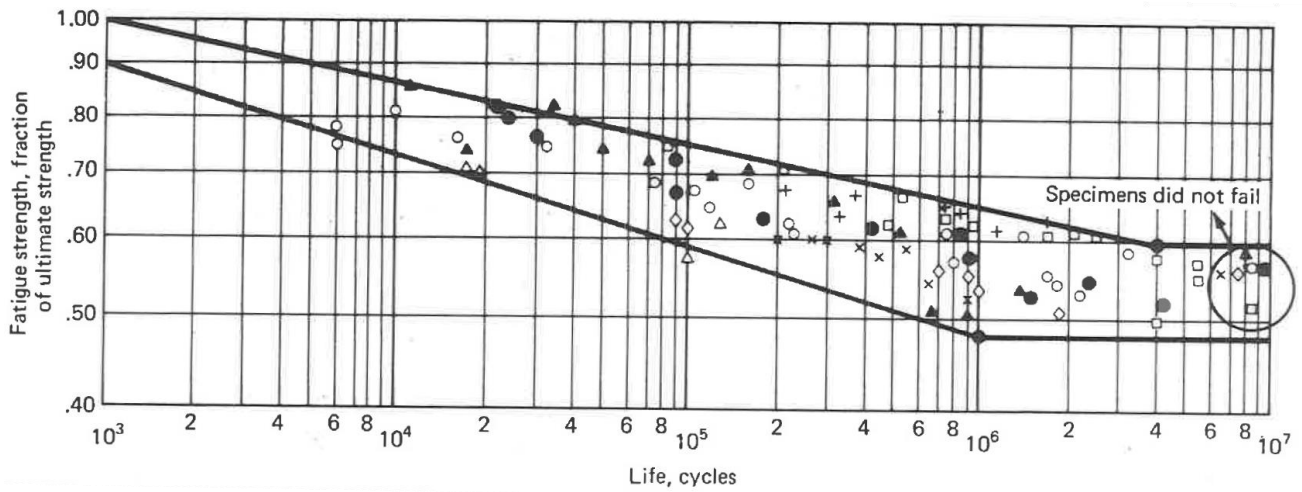


Figure 6 A non-dimensional steel bending stress versus loading cycles

If you do not have the S-N data for a material you can approximate the allowed stress by picking the expected number of cycles and using a conservative estimate of the S-N curve as shown below by assuming 50% allowed stress at a million cycles connecting to 90% at a thousand cycles:

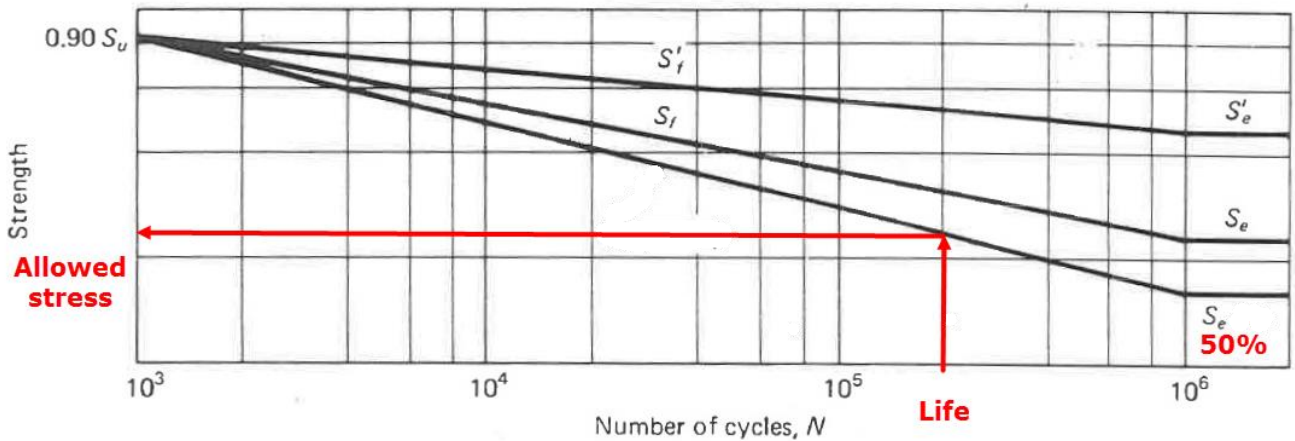


Figure 7 Estimating the allowed stress when no data are available

There are S-N curves for reversed bending loads, reversed torsion loads, and reversed axial loads. The torsion endurance limit is generally about 50% of the bending endurance as seen below:

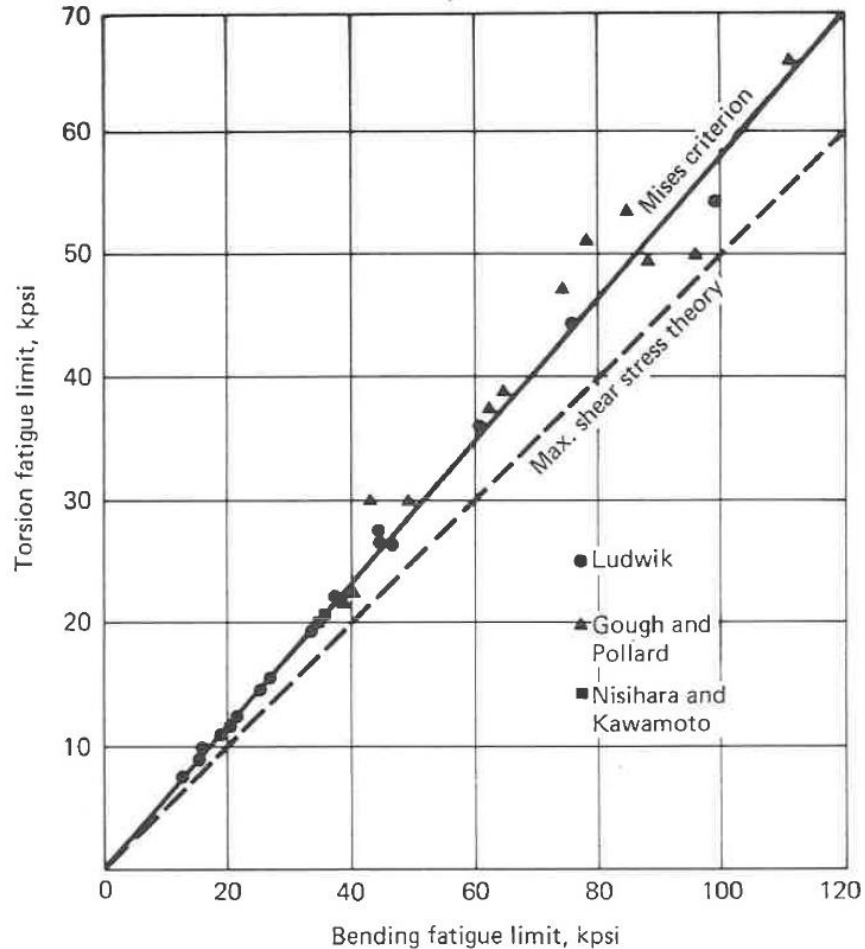


Figure 8 Torsional stress endurance is lower than bending stress endurance

- How to interpret natural frequency or buckling eigen-solutions? The magnitudes of the scalar eigenvalues are meaningful important numbers. (Zero values correspond to rigid body motion.) The eigenvector displacement magnitudes are essentially meaningless (and should not occur in the color bar)! However, their scaled values (from 0 to 1) give the important relative shape of the vibration or buckling mode. Most codes display only the scaled magnitudes.

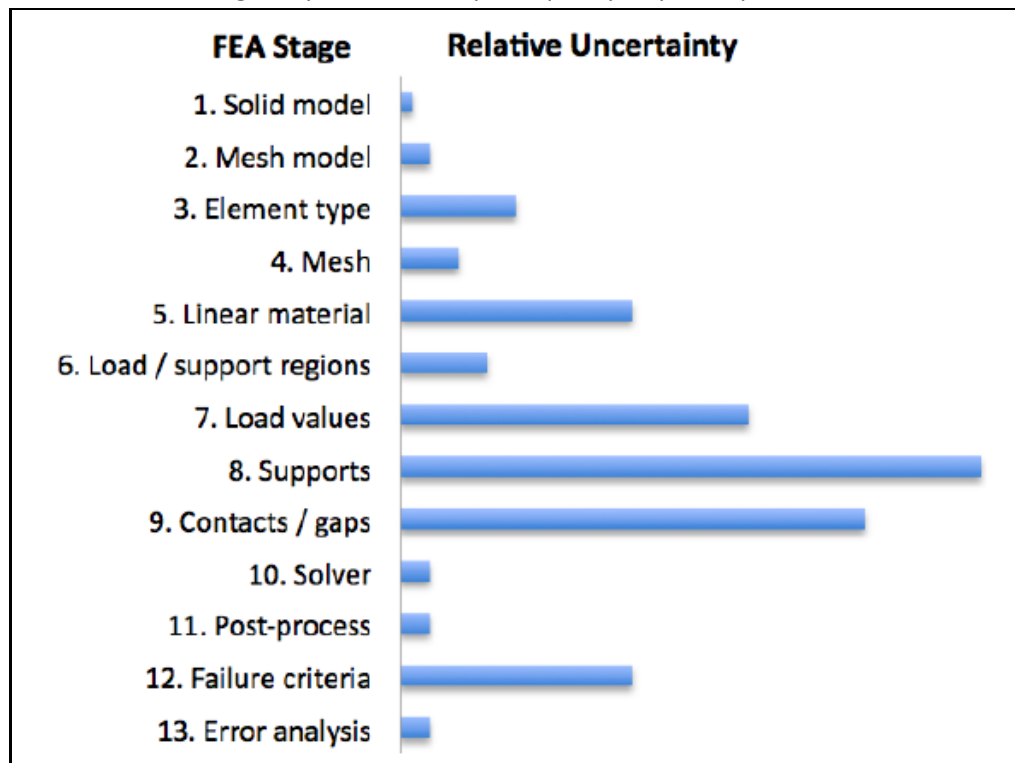
Buckling should always be checked for long skinny regions because it tends to give a sudden catastrophic failure. It is also easy to do with modern FEA. Buckling out of the plane of the loading is not unusual, so do not over restrain a buckling model. Of course, you must have enough restraints to prevent rigid body motion.

- If a body contains a sharp re-entrant corner, then a singularity occurs in the stresses and/or heat flux at such a location. In theory, they go to infinity at such a point and a FEA code can never reach that value. That is the main reason why most mechanical parts have fillets at re-entrant corners. In practice, the corner either has some small radius or inelastic behavior so the stress or heat flux does not go to infinity.
- The calculated Factor of Safety relates only to the state of the material. The overall Factor of Safety is obtained by multiplying that value by a group of values that are each greater than one.

Factors to consider when evaluating a design (each ≥ 1)

k	Type	Comments
1	Consequences	Will loss be okay, critical or fatal
2	Environment	Room-ambient or harsh chemicals present
3	Failure theory	Is a part clearly brittle, ductile, or unknown
4	Fatigue	Does the design experience more than ten cycles of use
5	Geometry of Part	Not uncertain, if from a CAD system
6	Geometry of Mesh	Defeaturing can introduce errors. Element sizes and location are important. Looking like the part is not enough.
7	Loading	Are loads precise or do they come from wave action, etc.
8	Material data	Is the material well known, or validated by tests
9	Reliability	Must the reliability of the design be high
10	Restraints	Designs are greatly influenced by assumed supports
11	Stresses	Was stress concentration considered, or shock loads

9. The results of a simulation are greatly influenced by the quality of your input data:



10. Use the proper number of significant figures. For example, if the modulus of elasticity is given to three significant figures, then the stress cannot be known to higher accuracy. Data are often miss-represented due to substitution into a units conversion equation.

Property	Value	Units
Elastic Modulus	2.1e+011	N/m ²
Poisson's Ratio	0.28	N/A
Shear Modulus	7.9e+010	N/m ²
Mass Density	7700	kg/m ³
Tensile Strength	723825600	N/m ²
Compressive Strength		N/m ²
Yield Strength	620422000	N/m ²

Property	Value	Units
Elastic Modulus	30457924.91	psi
Poisson's Ratio	0.28	N/A
Shear Modulus	11457981.28	psi
Density	0.27818	lb/in ³
Tensile Strength	104982.03	psi
Compressive Strength		psi

11. Review the basic concept of structural mechanics. Any structure or part can be thought of as a linear elastic spring. The spring can have applied loads and/or imposed displacements. In a simulation the spring stiffness(es) are obtained by integrating (numerically) quantities over the volume of the part or structure. Requiring the system to be in equilibrium yields a matrix relation between the displacements and the forces.

Specifying known displacement values (fixtures) reduces the matrix system to a size that can be solved for all remaining displacements. After all of the displacements are known the original matrix equilibrium equation can be solved for the reaction forces at the specified displacements. From the spatial gradient of the displacements we obtain the strains. From a material law the stresses are obtained from the strains. Once the stresses are known they are used to compute material failure laws.

The equilibrium relation for a simple spring between its stiffness, k , displacement, u , and axial force, f , is usually seen as: $k u = f$, or $u = f/k$. This simple form arises only because we initially assumed that one end of the spring was restrained from displacement (immovable), and the opposite end was subjected to a constant force. If the spring model is generalized to allow either end to be restrained or loaded then the equilibrium equation takes a matrix form:

$$k \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix}, \quad (1)$$

where the subscripts 1 and 2 refer to the left and right ends of the spring, respectively. Note that the determinant of the matrix is zero. That is because we must later add restraint information about at least one end to obtain a unique physical solution.

For example, assume that the left node has a known displacement, $u_1 = u_{given}$, (which is usually zero) and the right end has a known force, $f_2 = F$. Then, the unknowns are the right displacement, u_2 , and the left end reaction force, say $f_1 = R$. The revised analytic equilibrium relation is

$$k \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_{given} \\ u_2 \end{Bmatrix} = \begin{Bmatrix} R \\ F \end{Bmatrix},$$

and the independent displacement is found from the second row:

$$k[-u_{given} \quad u_2] = F$$

or

$$u_2 = u_{given} + F/k.$$

This is the same as the common form when u_{given} is zero, namely $u_2 = F/k$. Now the reaction force necessary to maintain u_{given} is obtained from the first row of the matrix system:

$$k[u_{given} \quad -(u_{given} + F/k)] = R$$

or simply $R = -F$, as expected. Any elastic structure behaves in a similar way. However, the stiffness matrix is obtained by numerically integrating its strain energy over the entire volume.