Linear vs. Non-Linear Contact Analysis

Purpose:

The purpose of this document is to demonstrate how to properly conduct Finite Element Analysis, FEA, on an Edge Rate style contact (Samtec's C-209-05.0-X was used). This document will also show the correlations between linear FEA, non-linear FEA, and actual test data.

Linear vs. Non-Linear FEA Analysis:

A linear FEA analysis is undertaken when a structure is expected to behave linearly, i.e. obeys Hook's Law. The stress is proportional to the strain, and the structure will return to its original configuration once the load has been removed. A structure is a load bearing member and can normally be classified as a bar, beam, column, or shaft.

Conversely, a non-linear FEA is used to predict the behavior of a structure that is loaded beyond the elastic limits of the material of interest. The structure experiences plastic deformation and will not return to its original configuration or shape.

Always perform a linear FEA on any structure whose behavior is unknown. A comparison of the Von Mises' stress level in structure and the yield strength of the choice material will determine if a non-linear FEA is required.

The results of a linear FEA will be accurate and acceptable if the proper boundary conditions and meshing have been applied. The Von Mises' stress level must be below that of the yield strength of the material. Remember, stress is a function of geometry only while non-linear behavior is a function of stress and material properties.

In the event that the Von Mises' stress level is greater than the yield strength of the chosen material, you have three options. First, perform a non-linear analysis to determine the degree of the plastic behavior and its suitability for the application. Second, look for a better material. Third, change your geometry to better manage the stress.

Changing boundary conditions and load stiffening may also dictate the use of a non-linear FEA, but they are beyond the scope of this particular analysis.

Stress-Strain Curve:

A Uniaxial Tensile Test is used to derive the material property know as Young's Modulus or the Modulus of Elasticity. Stress is calculated by dividing the load by the cross sectional area of the test specimen. Strain is calculated by dividing the change in length by the original length.

Every material will respond to the applied force in different ways generating different curves. The slope of the curve at any given point is the Modulus of Elasticity of the material. In the elastic region, the slope is generally linear, so the Modulus of Elasticity remains constant. For an example, the stress-strain curve for BeCu 390, a commonly used material, is shown in Figure 1.

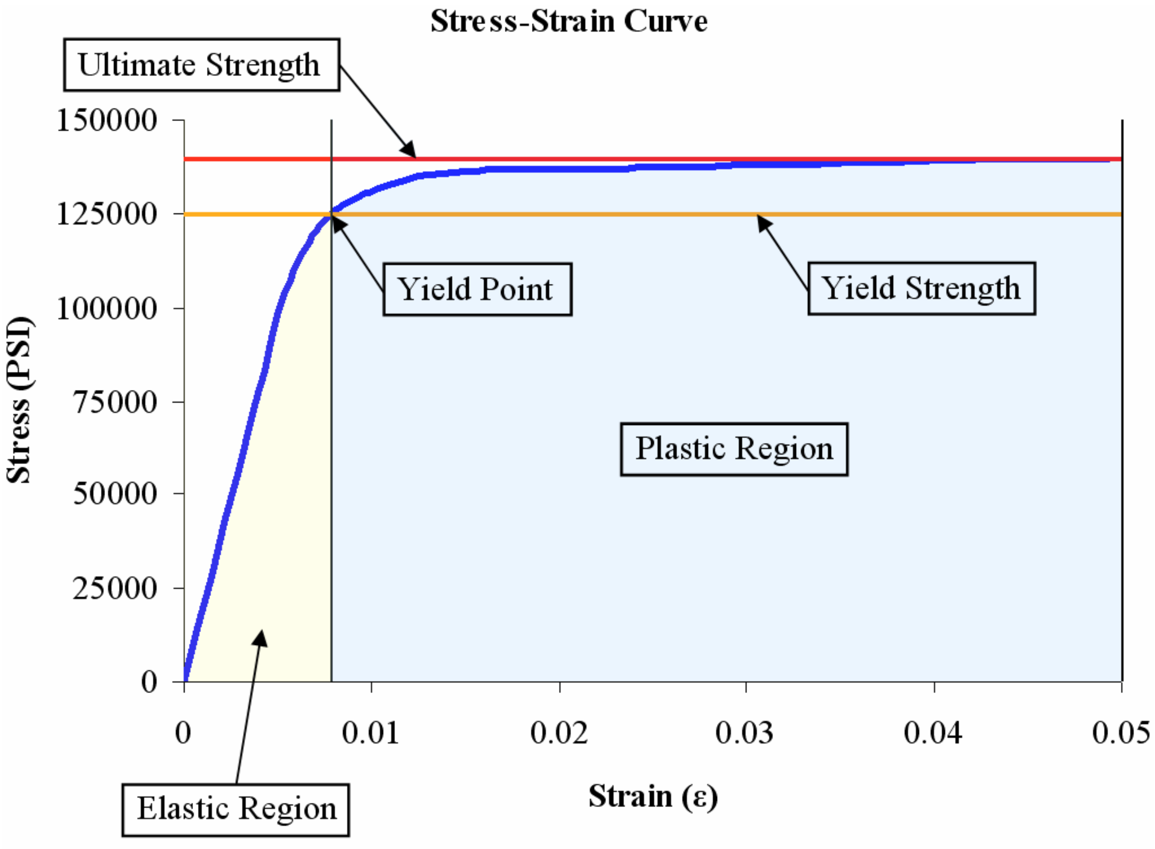

Figure 1. Stress-strain graph for BeCu 390.

As the figure shows, the material has a yield point around 125 KSI at a strain of 0.08. Once the material reaches the yield point, the contact will begin to show signs of plastic or permanent deformation and will not return to its original configuration. Set is an actual measure of the deformation.

Contact Normal Force:

In contact design, the normal force is an important factor in maintaining the proper connection between two contact mating surfaces. A force deflection curve is a plot of the normal force vs. the contact deflection. This graph will show how the contact will perform in a given deflection range. The stiffer the contact is designed, the steeper the normal force curve will be. The measured normal force curve for Samtec's C-209-05.0-X contact is shown in Figure 2.

Figure 2. C-209-05.0-X measured normal force versus deflection.

As Figure 2 shows, the normal force generates a working region which is the area in which the contact provides an adequate amount of normal force for its application. In a good design, this working region should be maximized to account for the variation in contact compression in customer applications. A large working region will also compensate for tolerances in the manufacturing of the connectors. The area of deflection beyond the working region shows an increase in the normal force slope because the contact begins to interfere with the connector body.

Contact Finite Element Analysis:

Finite Element Analysis, or FEA, is often used in the initial design stages of a new contact. This analysis will test the contact in the computer to generate a normal force curve based off the stress-strain curve provided by the material manufacturer. To obtain effective results from a FEA study, the restraints and contact conditions must be carefully laid out. Figure 3 shows the C-209-05.0-X contact with the proper restraints applied.

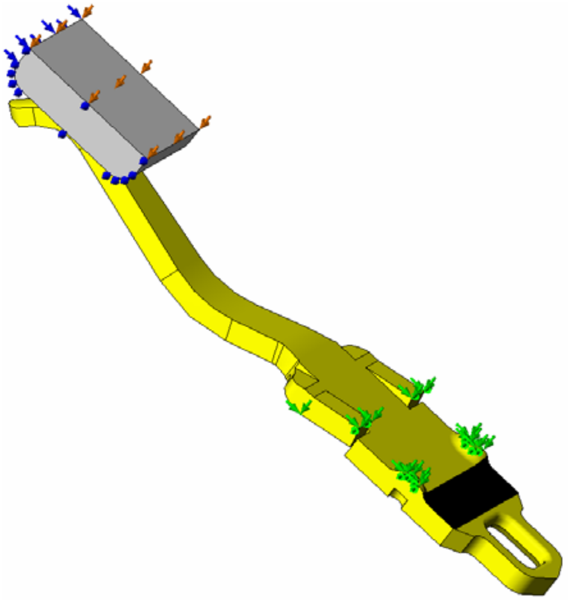

Figure 3. Contact restraint setup 1.

The green arrows in Figure 3 represent an immovable restraint that is applied to the areas on the contact that will be restrained when inserted into a body. The immovable restraint should be used on every contact because it prevents movement in the X, Y, and Z directions while still allowing rotation in any direction. This is a more accurate restraint method versus using a fixed option. The blue arrows are slider/roller restraints which are used to ensure that the contact pad being used stays normal to the contact when the displacement is applied. The orange arrows are the displacement restraints that are applied to the back side of the contact pad to simulate the force that is applied to the contact beam. These restraints are very important and must resemble the real world application as closely as possible to obtain adequate results.

Some other important items to set up prior to running the analysis are to ensure the material for both bodies are set. The contact material should be selected according to the desired choice, and the contact pad should be set to a different material that has a much harder surface to prevent the deflection of the pad's surface from having any effect on the results. Once the material is defined, the contacts/gaps should all be defined. The global contact set must be changed to no penetration to ensure that the pusher and contact are treated as separate entities. The next step is to define a contact set between the contact and contact pad surfaces to ensure there is no penetration and no gap between the two objects.

Once the contact is fully restrained and defined, the FEA can be run. Both linear and non-linear studies may be conducted, but it is important to keep in mind that the linear FEA does not account for any permanent set (or yielding) of the material. However, the linear analysis may be performed much quicker and can be used as a tool to check for any setup mistakes. Figure 4 and 5 below show the results from both a linear and non-linear analysis.

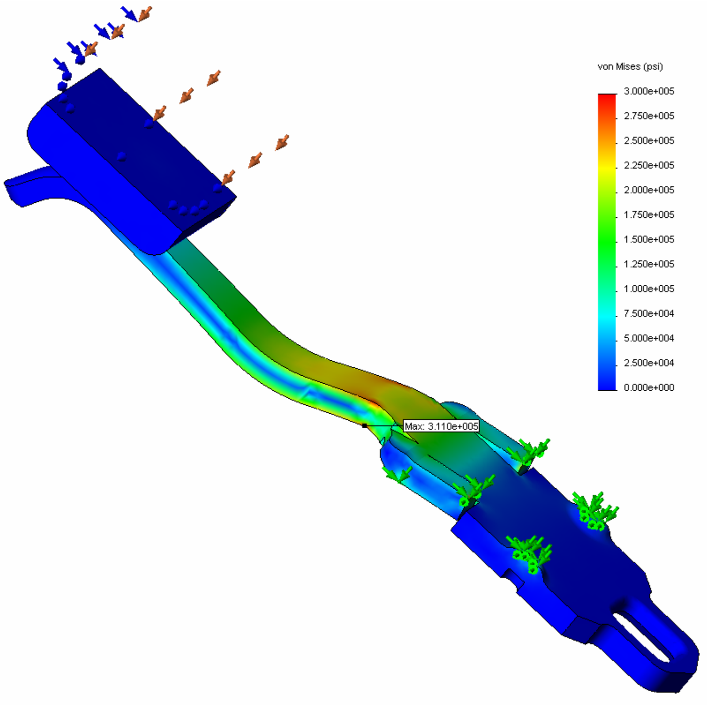

Figure 4. Linear FEA for C-209-05.0-X contact at 0.014" deflection.

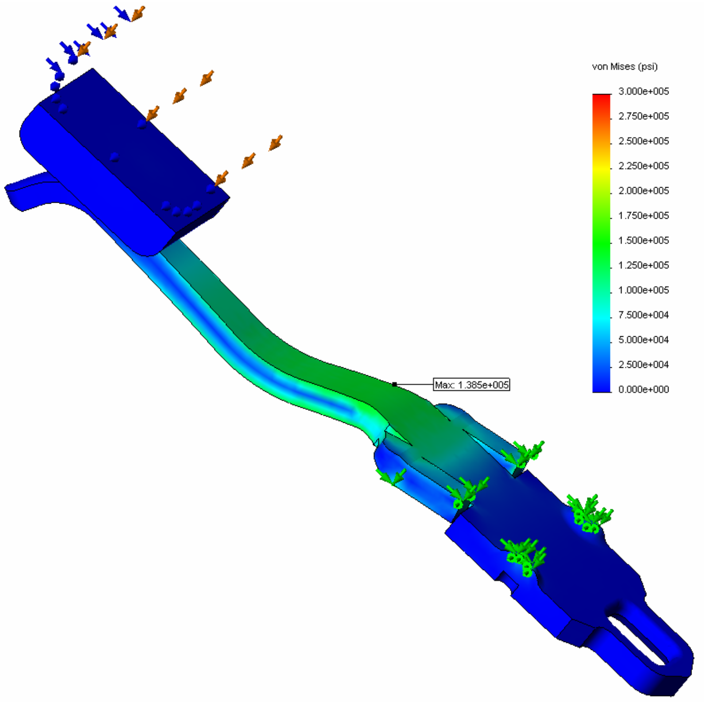

Figure 5. Non-Linear FEA for C-209-05.0-X contact at 0.014" deflection.

The FEA also records the reaction (or normal force) the contact generates as it is deflected. The generated values have distinct differences between the linear and non-linear FEA methods. A third, more basic method that can be used is a calculated reaction force value based on the size of the beam using

$$F_R = \frac{dEbh^3}{4L^3}$$

where d is maximum beam deflection, E is the modulus of elasticity, b is the beam width, h is the beam height, and L is the beam length. Figure 6 shows a graph of the normal force values generated by the linear FEA, non-linear FEA, and also the calculated beam reactions.

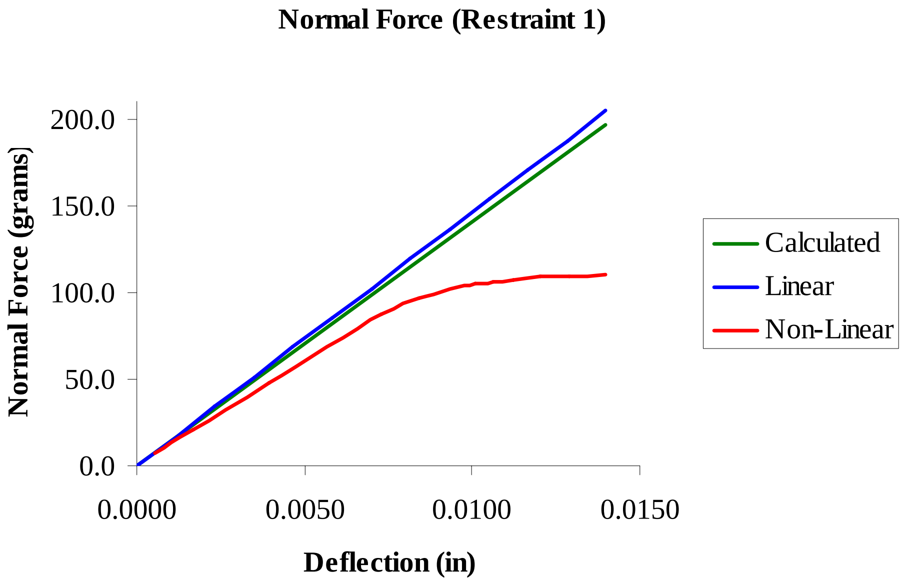

Figure 6. Normal Force vs. Deflection.

As Figure 6 shows, the linear FEA and calculated values of the normal force do not account for the yielding of the material. The non-linear analysis shows that after roughly 0.008” of deflection, the beam begins to yield and enters the plastic region. This yielding will cause permanent set and must be considered when designing the contact.

For an example, the restraints on the C-209-05.0-X contact were slightly modified, and the linear and non-linear FEA was regenerated. The restrained model is shown in Figure 7 where the same symbol scheme as before is used. The green arrows are the immovable restraints, the blue arrows are the slider/roller restraints, and the orange arrows are the displacement restraints. The same materials and displacement of 0.014” was used. The difference in this model is that the immovable contact restraints were changed from the points at which the contact will interfere with the body to restraining the entire front and back flat surfaces of the contact. This will affect the actual overall beam length which will increase the stress concentrations and increase the normal force.

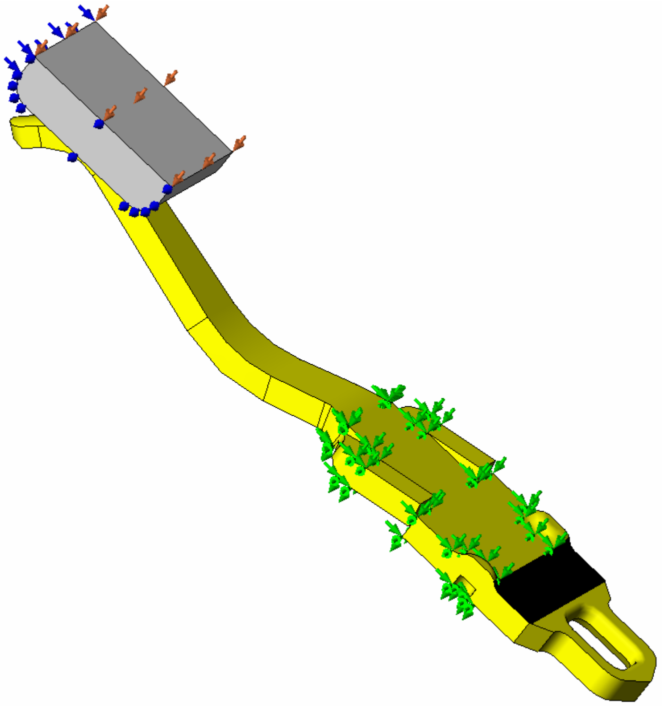

Figure 7. Contact restraint setup 2.

The results of the FEA are shown in Figure 8 and 9. The stresses generated with the modified restraints are slightly higher.

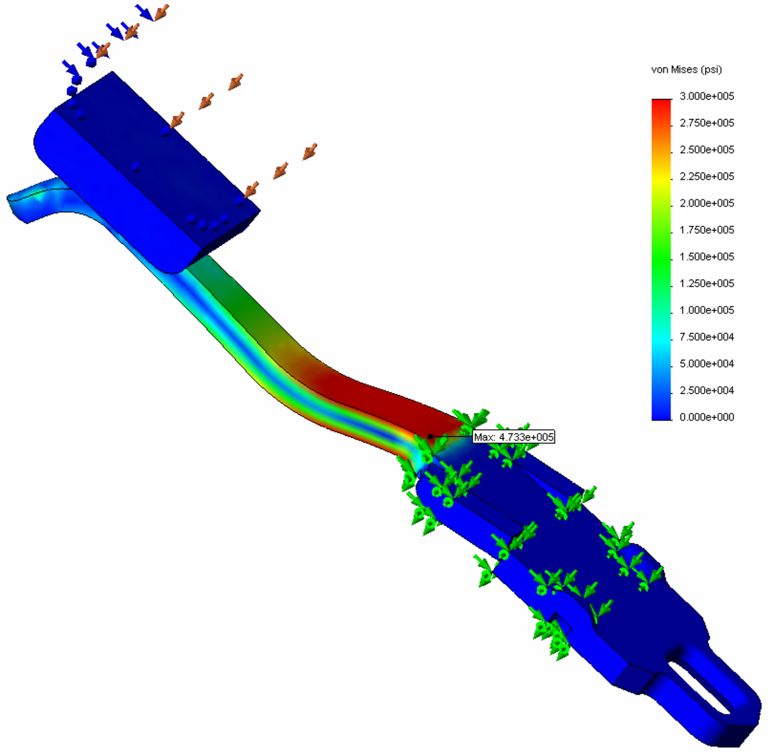

Figure 8. Linear FEA for C-209-05.0-X contact at 0.014” deflection.

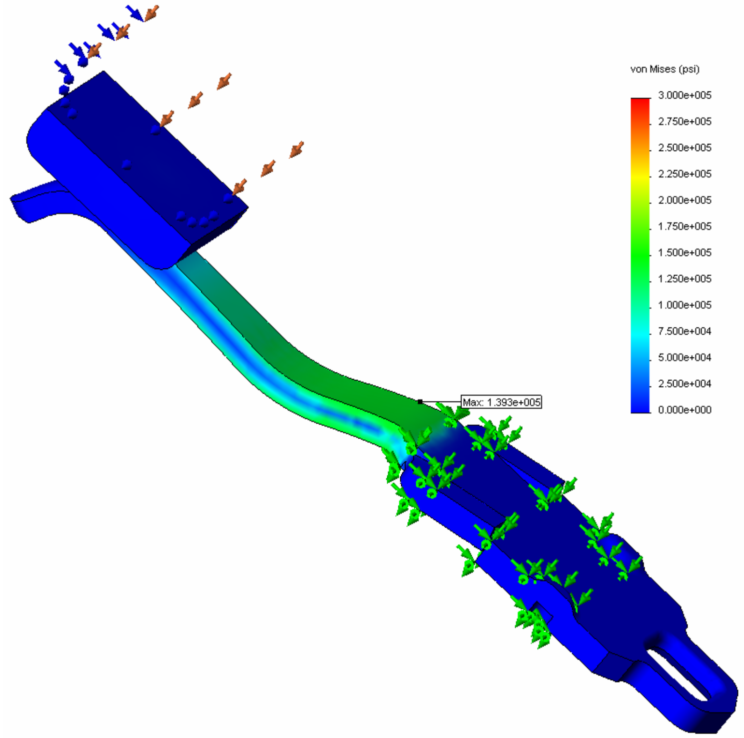

Figure 9. Non-Linear FEA for C-209-05.0-X contact at 0.014" deflection.

With the modified restraints, the normal force readings were also increased as shown in Figure 10.

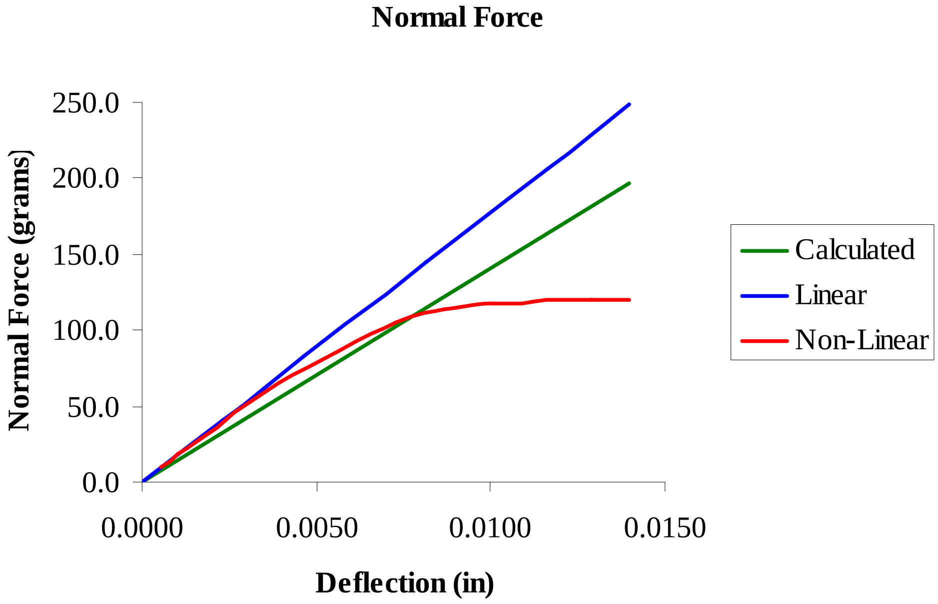

Figure 10. Normal Force vs. Deflection for the 2^{nd} set of restraints.

Conclusion:

Finite Element Analysis is a very effective tool in designing and analytically testing a contact in the initial stages of development. The correlations between a proper FEA study and finished product has proven to be similar; however, slight mistakes or changes in a FEA model can skew the results. The analysis type (linear and non-linear), restraints, loads, and material selections must be closely monitored to achieve desirable results.

Authors:

Brian Vicich; Craig Ryan; Kevin Meredith