Available online at http://www.journalijdr.com

International Journal of DEVELOPMENT RESEARCH

ISSN: 2230-9926

International Journal of Development Research
Vol. 06, Issue, 12, pp.10527-10533, December, 2016

Full Length Research Article

ELASTIC AND ELASTO-PLASTIC FINITE ELEMENT ANALYSIS OF A TENSION TEST SPECIMEN WITH AND WITHOUT VOIDS

***Ravindranath Tagore, K. and Sidda Basha, D.**

Department of Mechanical Engineering, EIET, Hyderabad-502319, India

ARTICLE INFO

Article History:
Received 25th September, 2016
Received in revised form
22nd October, 2016
Accepted 29th November, 2016
Published online 30th December, 2016

Key Words:
Anhydrous ammonia;
Anhydrous ammonia
Application equipment;
Sunflower;
Fertilizing application;
Nitrogen.

ABSTRACT

Finite Element Analysis (FEA) is an important engineering tool used to assist in approximating and verifying how a component will react under various external and internal loading conditions. The purpose of this master of engineering project is to investigate and analyze fully elastic and elastic-plastic deformation of a High Strength Steel (HSS) tensile test specimen in FEA ANSYS under various conditions. In general, material selection is a key component to engineering analysis. If the material selection is performed incorrectly and does not have the strength, ductility, or physical features to withstand the load then failure will become a realistic result. There are many conditions where the stress applied could extend beyond the yield point of the material but FEA does not necessarily provide visual indication that the part has yielded. The initial results showed that the fully elastic material property has a linear stress-strain relationship regardless of the load applied while the elastic-plastic has a linear relationship up to yield point and then becomes nonlinear beyond yield point. This FEA investigation will also include elastic-plastic analysis on reverse loading, cavity geometry, and random pores within the HSS tensile test specimen. This will be accomplished by using different modeling techniques to investigate how FEA ANSYS analyzes elastic-plastic material deformation under various loading conditions and material conditions. The reverse loading condition resulted in tensile and compressive stresses that were equal and opposite even in the plastic range. Work hardening should have increased the strength of the test specimen so the compressive stress should have been smaller than the tensile loading. The plastic data from the tensile test was not cyclic and only applied tensile force so FEA ANSYS did not know how to interpret the plastic range for compression.FEA ANSYS simply applied a negative force which resulted in an equal and opposite stress. The cavity model had higher stress than the tensile and reverse loading conditions with the elastic-plastic data. FEA showed significant visual deformation within the cavity of the specimen. The high stress and deformation were due to the reduction in cross sectional area because stress is a function of force over the area. The pores modeled within the test specimen had minimal effect on the overall stress, strain or deflection compared to the theoretical HSS test specimen that had no imperfections. The pore had high stress because the pores were a source of high stress concentrations but the stress and strains not within the pores did not change. The fully elastic and elastic-plastic material selections each play a vital role in FEA ANSYS while both can contain their sources of errors. In the fully elastic model, regardless of the amount of stress applied, the stress-strain curve acted linear for the fully elastic properties even beyond yield point. The elastic-plastic model applies actual stress strain test data beyond the yield point and applies it to the part being analyzed. The results showed nonlinear stress but when stresses outside the uploaded values were applied FEA ANSYS analysis resorted back to a linear stress-strain relationship.

INTRODUCTION

The objective of this master of engineering final project is to investigate and analyze Finite Element Analysis (FEA) ANSYS when it is used to evaluate fully elastic and elastic-plastic deformation on a High Strength Steel (HSS) tensile test specimen. A tensile test specimen will be modeled in FEA ANSYS as if it was being loaded in a tensile test apparatus to validate the material properties. The objective of this analysis is to obtain a better understanding how FEA ANSYS analyzes fully elastic and elastic-plastic material properties under various loading and material conditions. By placing different

****Corresponding author: Ravindranath Tagore, K.***
Department of Mechanical Engineering, EIET, Hyderabad-502319, India

conditions on a simple geometry (Cavity and Pores analysis), it is easier to obtain a better understanding how FEA ANSYS analyzes the different models and how changing conditions affects the overall strength of the component. The objective of the cavity is to gain more knowledge on how the HSS test specimen will react to a different internal geometry. The objective of the pores analysis is to verify the overall strength of the HSS test specimen when imperfections are present within thematerial. One of the major topics within Finite Element Analysis (FEA) is how to interpreter results and verify whether the results accurately depict the loading condition. Inaccuracies within the FEA results can stem from various sources including modeling, input of material properties (fully elastic, elastic-plastic, thermal, fluid, etc.), mesh density, unrealistic stress concentrations, loading conditions, environment, etc.

Engineers understand FEA is an approximation tool and the analysis has to be validated by hand calculations, test results, or inspection to confirm the validity of the results. Hand calculations are sufficient for calculating theoretical solutions for a piece part component under a single or simple loading condition. But as the design increases in complexity, as various loads are applied, and as the environment complexity increases, the engineer requires assurance that the FEA results are correct. Multiple loading and environment conditions make hand calculations complex, where multiple assumptions have to be made to simplify the solution which could lead to inaccurate results. The more assumptions made will lead to questionable results. Material selection and understanding how material properties interact within FEA ANSYS is a major factor in confirming the results. This project will analyze the use offully elastic material properties in FEA ANSYS and elastic-plastic material properties to investigate how FEA ANSYS analyzes fully elastic and elastic-plastic deformation under a high loading condition of High Strength Steel (HSS).

The predicted FEA results will look like Figure 1 that shows a stress-strain curve for HSS from actual test data. Hand calculations are used to verify the linear geometry to obtain a better understanding how hand calculations compare to FEA results. Figure 1 depicts the fully elastic range and the transition into the plastic range for HSS from lab testing. Another method to validate the FEA results is by hand calculations. Hand calculations are simpler in the elastic range because of the linear relationships that can be developed between stress, strains, and deflections. Hand calculations can be used in the plastic range but nonlinear analysis is not simple and typically requires numerous assumptions. Material that has dislocations demonstrates that the bonds will slip and the specimen will Hooke’s law is a linear relationship that relates stress to strain by using the modulus of elasticity of the material [4]. Fully elastic analysis only pertains to stress and strain up to the yield point. Past yield point, the HSS should start to plastically deform. In FEA, to analyze items within the fully elastic range only the modulus of elasticity and material density are required to perform a structural analysis.

The analysis is assumed to be accurate up to yield point but will continue to act linearly beyond the yield point. The linear relationship past the yield point will result in an inaccurate result for high stresses and high strains. Prior to looking at the physical defects of elastic-plastic analysis, it is important to review the plastic stress-strain relationship at the atomic / crystal level. Plasticity in metals is usually a consequence of dislocations within the structure [5]. Plasticity will occur beyond the yield point of the material. For HSS the yield point is 54,000 psi which is found in MIL-S-22698, Rev C, and condition AH-36. For many ductile metals, including HSS, tensile loading applied to a test specimen will cause it to behave in an elastic manner [6]. Once the load exceeds the yield strength threshold, there is more of a rapid increase in stress than in the elastic range, and when the load is removed some amount of the extensions still exists which results in

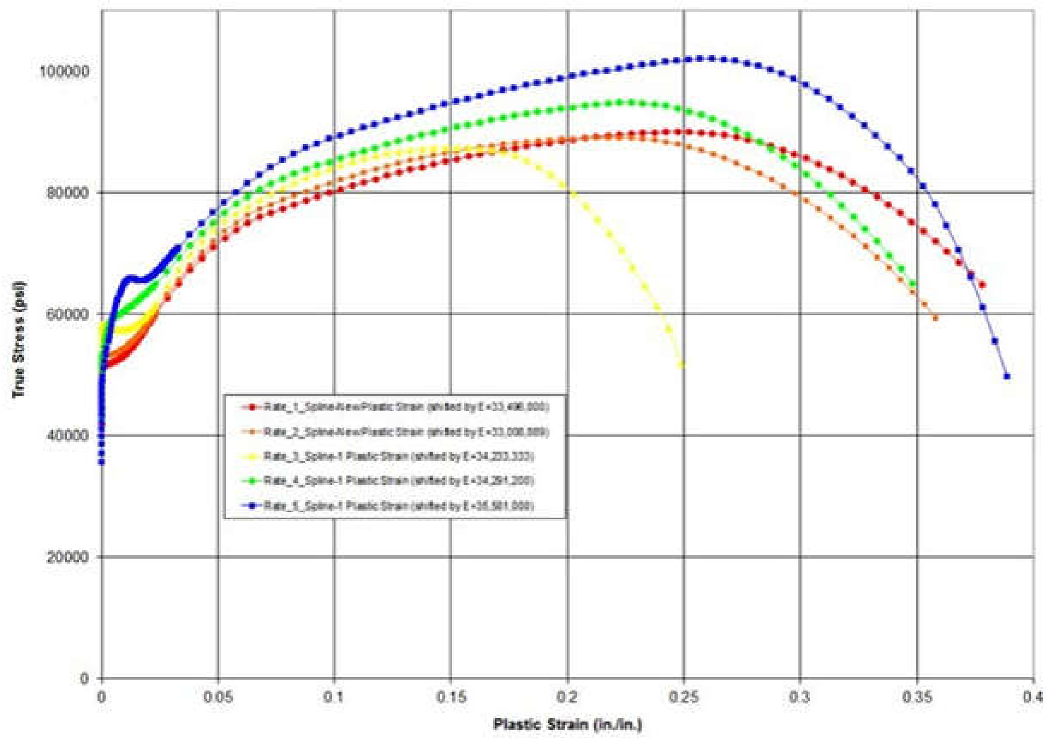

Figure 1. Engineering stress vs Engineering strain

PLASTIC deformation in practice typically means the component was under designed and was not capable of handling the environment for which it was meant to operate in. This will require additional rework or in a worse case scenario a complete redesign to ensure that the component does not plastically deform and still has enough strength to support the operational loading conditions. A change in material to increase the strength would be the simplest solution to this problem but the engineer must take into consideration availability of the metal, cost, and the other physical properties such as ductility, brittle- ness, ultimate strengths, etc. There is cost increase associated with both redesignoptions. Plastic deformation can be beneficial as when used to produce work strain hardening by metal working, also known as strain hardening or cold working [7]. Work hardening increases the number of dislocations [7] forming new, stronger crystal bonds. Energy is almost always applied fast enough and in large enough magnitude to not only move existing dislocations, but also produce a great number of new dislocations by jarring or working the material sufficiently enough [7].

MATERIALS AND METHODS

Methodology

The method for investigating the various materials and loading conditions is to model a HSS test specimen in FEA ANSYS. Once modeled the test specimen will be put under various material conditions and loading. The first portion of the investigation will involve a HSS tensile test sample, modeling it in ANSYS, and applying a large tensile load on the sample. The first analysis will only focus on fully elastic material properties up to the yield point and then investigate how FEA performs with large loads beyond the elastic range.

simulation, and pores analysis. Once the analysis is complete, a comprehensive comparison will be performed to derive a conclusion on the fully elastic versus elastic-plastic deformation in FEA.

Modeling: Created Test Specimen in FEA

Perhaps one of the most important tests performed on a material is the tensile test. In a tensile test, one end of a rod is clamped in a loading frame while the other end is subjected to a controlled displacement [9]. A transducer is typically connected in series with the specimen to provide an electronic reading of the load corresponding to the displacement [9]. The analysis included in this report will model a cylindrical test specimen from ASTM E8. Test specimens 1 from ASME E8 were then modeled into FEA ANSYS. Because this is a cylindrical specimen, the revolve function was used in FEA. Half of the longitudinal cross section was modeled and revolved around the y-axis (centerline). With the sketch complete, the part is revolved around the centerline to develop the final solid shape that will be analyzed as shown in Figure 2. Radii have been applied to all sharp corners because sharp corners on any component are a source of high stress concentrations. To alleviate this, radii are applied to the edges and this will provide a more accurate result.

Material Properties

The material properties for High Strength Steel (HSS) came from MIL-S-22698, Revision C, and condition AH-36. Table 1 provides the mechanical properties found in MIL- S-22698: To perform the fully elastic analysis one material had to be created. This material has three physical properties that need to be applied to the test specimen which are Young's Modulus, Density, and Poisson's Ratio.

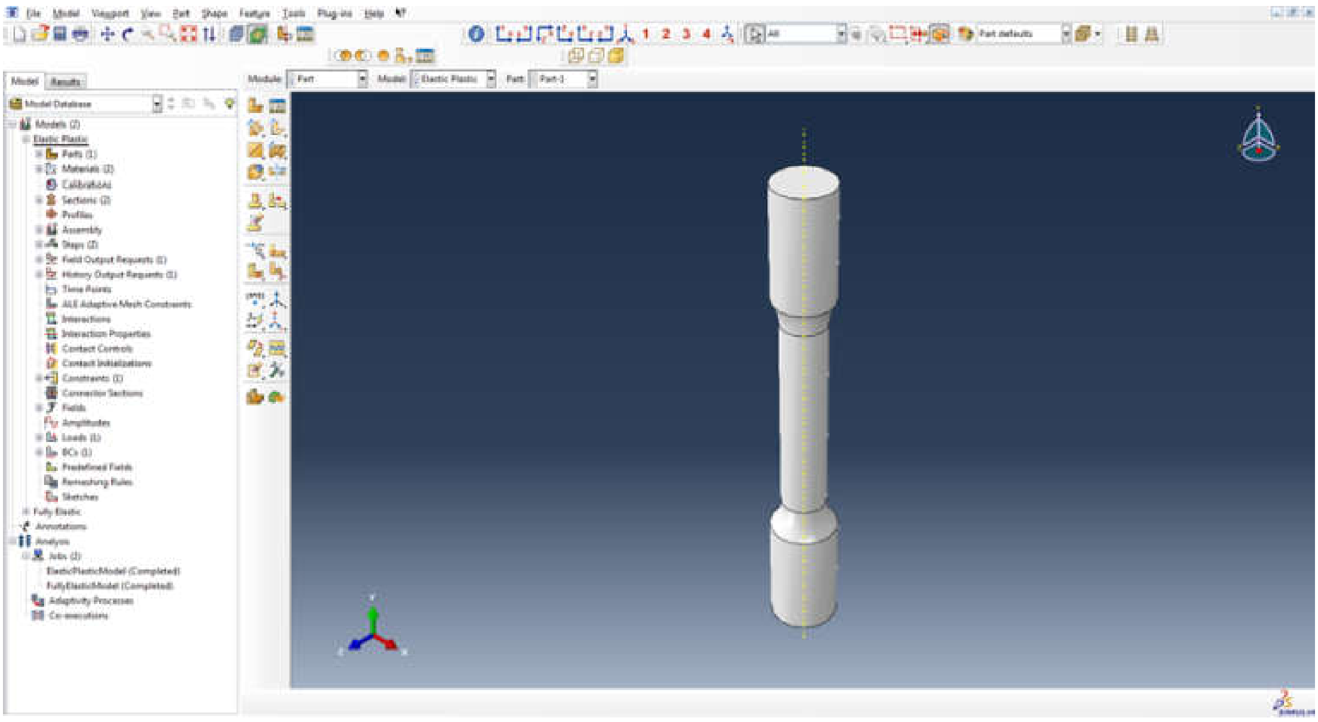

Figure 2. Created part revolved around center line

The second portion of this investigation will be to use the elastic-plastic material properties to perform the same tensile test under various conditions including reverse loading, cavity

To perform the elastic-plastic analysis, the elastic mechanical properties are still required because this will provide the stress-strain relationship up to the yield point but additional information is required to ensure the plastic range is accurately developed. To develop the plastic range in FEA, the yield stress and plastic strain need to be manually loaded into the material properties. Figure 3shows a small portion of the yield stress and plastic strain that was manually loaded from the excel file from Figure 1. To perform this analysis 283 yield stresses and plastic strains points were manually loaded into the plastic material properties module. This is sufficient because of the small change between each stress-strainpoint.

Table 1. Mechanical Properties MIL-S-22698

Tensile Strength	71,000 – 90,000 psi
Yield Strength (min)	51,000 psi
Density	0.283 lb/in^3
Poisson's ratio	0.3
Young's modulus	29,500,000 psi

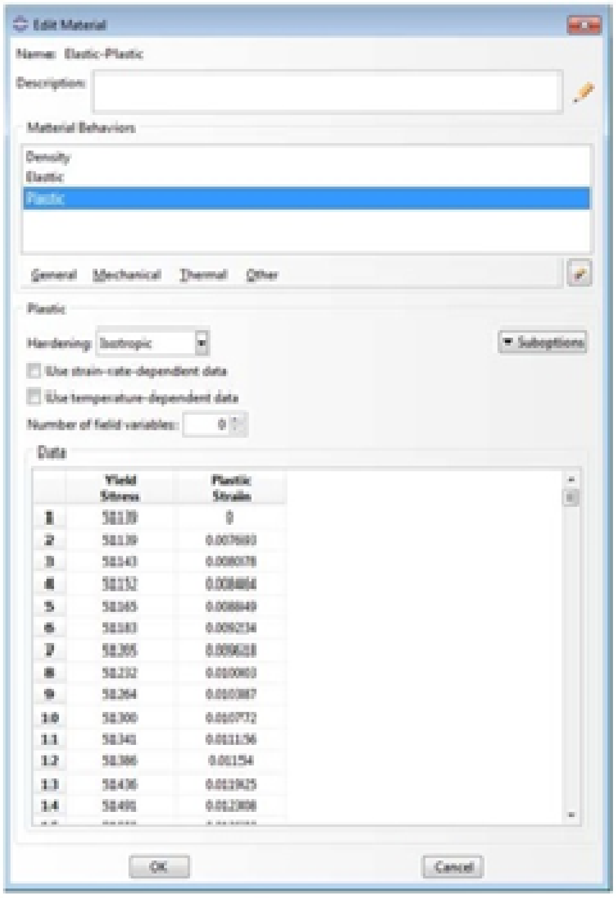

Figure 3. Yield Stress - Plastic Strain

Mesh

The mesh of the HSS test specimen is an important element to modeling the test specimen because a poorly meshed component could show stresses that are not realistic. If the elements are too large then the stresses between each element can be magnified butifthe mesh is too fine then the part might take up too much memory space within the computer and will take too long to run. A really fine mesh will not produce a more accurate result than if the mesh density was correct. A mesh density study was performed by changing the size of each element, running the analysis and viewing the results to see if the stresses changed with the change in seed size. This was done until there was no significant change in stress. The final element size for meshing the test specimen is 0.15 inches.

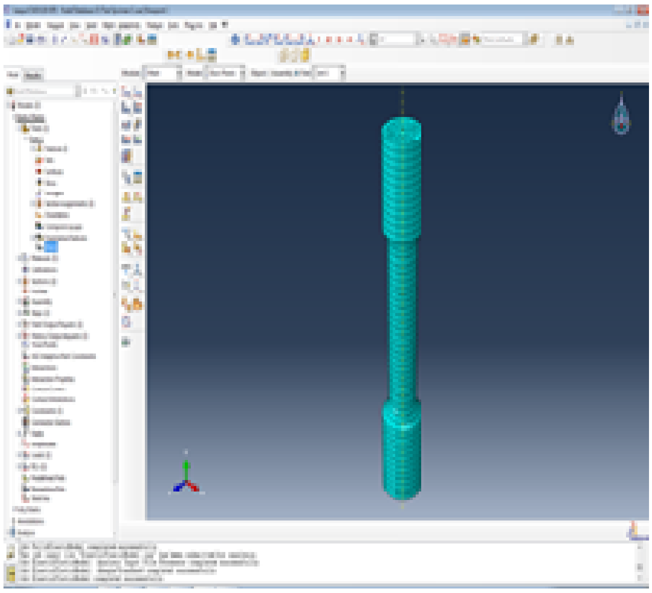

Figure 4. Mesh

Boundary Conditions and Loading

To best simulate the tensile test apparatus, one end of the test specimen is fixed in the test fixture and does not move. The other end of the test specimen is slowly pulled in tension to simulate the stress-strain curve of the HSS metal. For FEA to simulate this test, one end of the test specimen is fixed from displacing or rotating in the x, y, and z direction. This simulates the test specimen being threaded into the test apparatus where it would not be able to deflect, rotate, or translate. The boundary conditions are applied that show the end of the test specimen is fixed from displacement and rotation.

Calculating the required force to bring the High Strength Steel well beyond yield

$\sigma := 51000 \frac{lbf}{in^2}$ σ = yield stress from table 1

Calculating Area

$d := 0.5in$ d = diameter of test specimen neck

$a := \frac{\pi}{4} \cdot d^2$

$a = 0.196 in^2$

Calculating force

$f := \sigma \cdot a$

$f = 10014 lbf$ Minimum force required to yield part

Selected 15,000 lbf for analysis

A tensile load is then applied to the opposite end of the test specimen. To apply the load, a reference was created at the top surface of the test specimen. A reference point provides a point on the surface of the test specimen to apply the load. The load was then applied to the reference point and pulled a positive 15,000 lbf in the y-direction. The load was selected at 15,000 lbf because the calculate force to yield the test specimen was 10,014 lbf. To ensure test specimen goes well beyond the yield stress, a load of 15,000 lbf was applied. This same force can be applied to the fully elastic and elastic-plastic model to determine how ANSYS analyzes these results. The minimum force is calculated next and is a screen shot from MATHCAD.

RESULTS

FEA Results

The FEA results include Von Mises stress, displacement, and stress-strain curves from the FEA ANSYS program. The results will include analysis for the fully elastic tensile loading condition, elastic-plastic material properties under the various loading conditions. Figure 5 is the Von Mises stress as calculated in FEA ANSYS. As shown in Figure 5, there is high stress in the center of the test specimen while the highest stress is due to a notch which is due to the change in section areas. Stress is a function of force over the area so as the area is increased the stress will decrease. The maximum stress is located towards the 0.375 fillet and is not located at the center of the test specimen.

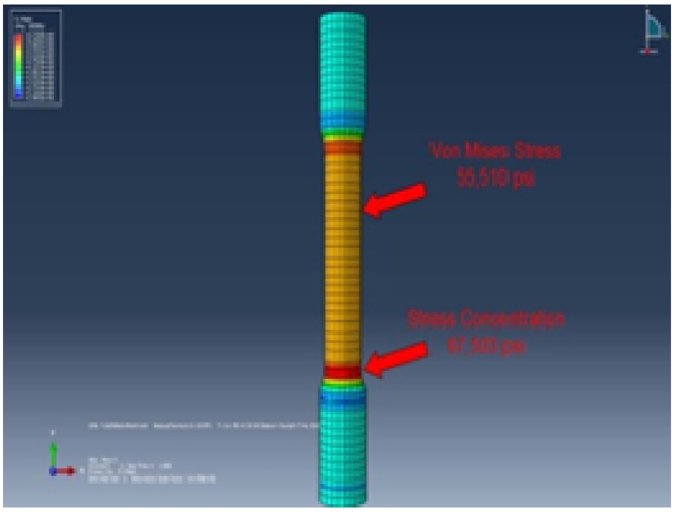

Figure 5. Von Mises Stress Figure

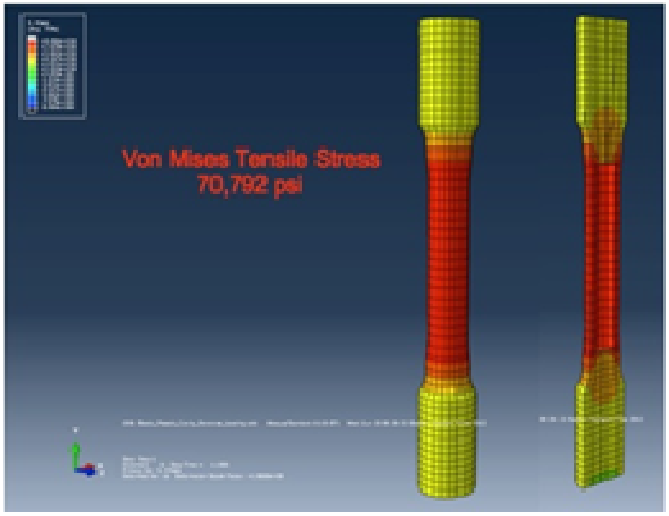

Figure 6. Reverse Loading under the Tensile Load

The calculated Von Mises stress is slightly less than FEA ANSYS Von Mises stress. The stress at the center at the test specimen according to the hand calculation is 54,019 psi. When comparing the calculated Von Mises stress to the ANSYS Von- Mises stress at the center there is an error of 2.7 percent. This is fairly accurate and the hand calculation validates the stress at the center of the test specimen. But ANSYS is showing a maximum stress of 67,503 psi. This is caused by a notch effect from the changing of areas.

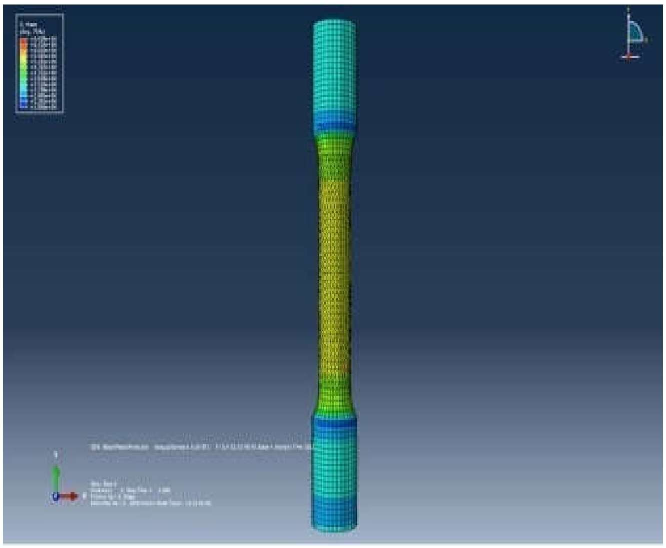

Figure 7. Stress with pores

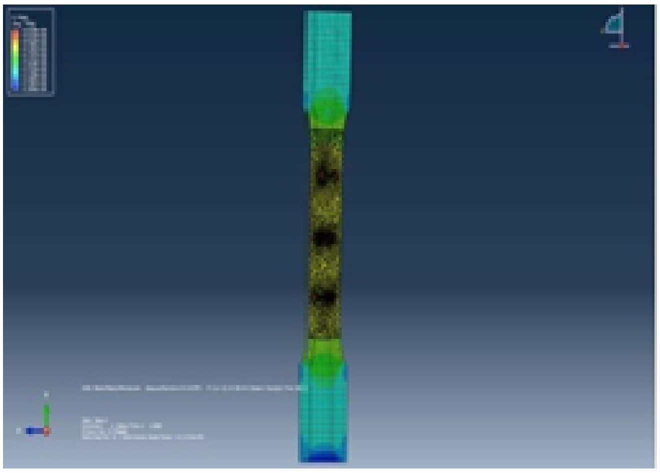

Figure 8. Cross Section Cut of Stress

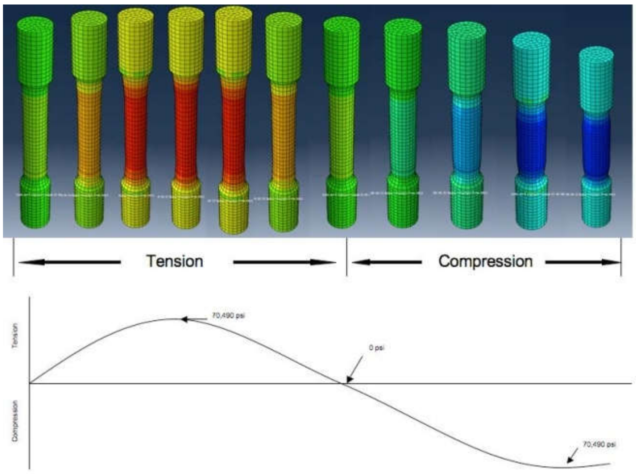

Figure 9. Reverse Loading: Cavity test specimen

Whenever there is a change in area along a continuous part, there are areas of stress concentrations. The stress concentration factor, k_t is calculated from Reference [10]. Any discontinuity in a machine part (i.e. fillets) alters the stress distribution in the neighborhood of the discontinuity so that the elementary stress equations no longer describe the state of stress in that part of these locations [10]. The next set of equations from Reference [10] calculates out the stress concentration factor, k_t, then applies that to the Von Mises calculatedvalue. With a stress concentration factor of 1.25 applied to the calculated Von Mises stress the high stress in this part is 67,524 psi while the FEA ANSYS Von Mises stress was 67,507 psi. This is a percent error of approximately 0.03 percent. If the k_t value went to more decimal places and was more exact, the percent error would be slightly smaller. The forces in all three directions applied. Considering this is only a uni-axial loading condition, there are small stresses in the z-axis and x-axis that do not contribute to calculating the Von Mises stress in FEA ANSYS. The small stress is due to the nodal bonds between each element. The link transmits load in all three direction and applies a small load in areas where force may not be acting in that direction. This is a source of error between the hand calculations and ANSYS. The maximum deflection was taken from the center of the test specimen that is 0.00836 inches. The highest deflection is at the end of the test specimen that is pulled with the tensile force. The portion with no deflection has all of the boundary conditions applied that keeps the specimen from rotating or deforming at the base.

DISCUSSION

This discussion section will further investigate the FEA ANSYS analysis for fully elastic and elastic-plastic material properties for a HSS tensile test specimen. Simple calculations verified the fully elastic material properties while actual test data ensured accurate results in the elastic-plastic range for tensile loading. The reverse loading condition with the elastic plastic material properties is an investigative study to analyze how FEA analyzes a component being extended into the plastic range then immediately. Compressed. Additionally the cavity and pores models demonstrated that the cavity has significant effects on the stress-strain of the test specimen while the pores model showed no change in the overall stress-strain that would cause the part to fail.

Fully Elastic FEA Discussion

As shown in the FEA results of Figure 5, the stress-strain relationship is linear regard- less of the amount of force is applied. FEA accuracy is based on the amount of fidelity incorporated into the design. By only adding material properties that pertain to elastic mechanical properties, the model will react linearly, even beyond the yield point. The hand calculations show there is a 2.9% variance in the Von Mises stress when compared to the FEA and a deflection of 9.5% different. Calculating the percentile difference in strength and deflection past the yield point is not practical because the test specimen going into the plastic range and nonlinearly deforming. Linear stress as a function of strain is not applicable in the plastic region and should be discarded in an engineering analysis. When comparing the stresses in ANSYS, two sections of the HSS test specimen were reviewed. The center section of the test specimen because it had a smaller cross section- al area and the fillet radius because this is an area identified by FEA as having a high concentration of stress. The smaller cross section of the test specimen shows the FEA ANSYS model and the hand calculations are consistent with one another. There is a 2.9% difference in strength with the hand calculations which is fairly consistent with each other. The stress concentration around the fillet is an expected location for a high stress but not a predicted failure mode. The failure mode seems to be within the center of the test specimen and is not around the fillet. When performing the elastic- plastic deformation the highest Von Mises stress was located at the center of the test specimen. Fully elastic material properties did not allow for realist failure results beyond the yield point and high stresses were shown in the wrong locations.

Conclusion

FEA ANSYS is an approximation tool that provides representative values to complicated problems. FEA is only as accurate as the fidelity implemented by the operator. The full elastic model showed a slight variation in stress and elongation, 2.3% and 9.5%respectfully when compared to hand calculations. The fully elastic stress-strain curve is a linear relationship which is not accurate past yield point. The elastic-plastic material properties showed that for pure tension, the plastic strains and stress governs the shape of the stress-strain curve. Also, the Von Mises stress and elongation was reduced when compared to the fully elastic analysis because of the nonlinear properties. The reverse loading of the elastic-plastic material properties demonstrated nonlinear deformation through the tensile loading condition but under the compressive load, the stress-strain relationship was linear and never fell below zero stress. Two additional cases were analyzed with the elastic-plastic material properties. The first analysis included a large cavity within the center of the test specimen. The cavity showed extremely high stress due to the reduction in area. The stress-strain curve was equivalent to the plot with no cavity except for the increase in stress values. The second investigation included the HSS test specimen with random pores imbedded within the body of the specimen that represented imperfections. The stress, strain and elongation did not change for the part and the only significant increase in stress was within the pores. The pores showed visual elongation but the stress-strain curve was the same as plot with no pores. The few pores did not affect the overall strength of the test specimen. This investigation demonstrated FEA ANSYS's ability to solve problems using elastic and elastic-plastic material properties. In general, the results showed similar results as that predicted but certain cases in the plastic range; especially compression in the reverse loading did not react as expected.

REFERENCES

Van Hengel, C. 2001. "Fiber Metal Laminates: An Introduction", Kluwer Academic Publisher, Volume 1, Issue 1, pp 15-16

Chao Isaac, "A simple geometric model for elastic deformations", Caltech, TU Munchen Germany, Volume 29, Issue 4

George E. Dieter, 1986. "Mechanical Metallurgy", McGraw-Hill Inc, New York New York.

Karlsruhe – Allemagne, 2003. "Tensile, Charpy, Creep and Structural Tests", vol. 6911, Issue 1, pp 5

Yang, Qiang, “Mechanical basis and engineering significance of deformation rein- forcement theory”, State Key Laboratory of Hydroscience and Hydraulic Engineering,

Tsinghua University, Bejing 100084, China, 2008

Batdorf, S.B. Budiansky, Bernard. 1980. “A mathematical Theory of Plasticity Based on the Concept of Slip”, Science Reviews 2000 Ltd, London UK.

Gil Sevillan, “Large Strain Work Hardening and Texture, Progress in Materials Science” Volume 24, Issue 2, pp 87-89

Robert D. Cook. “Concepts and Applications of Finite Element Analysis”, New York New York, University of Wisconsin – Madison, 1974

David Roylance, “Stress-Strain Curves”, Massachusetts Institute of Technology, Cambridge 2011

Joseph E. Shigley, 2004. “Mechanical Engineering Design”, Seventh Edition, McGraw- Hill, New York New York.

****