Finite Elements Analysis of Weldments with H- and P-Elements of Differing Orders of Interpolating Polynomials.

A.T. Oyelami, Ph.D.*; A.K. Ogunkoya, M.Eng.; O.D. Ogundare, M.Eng.; and B.A. Olunlade, Ph.D.

Engineering Materials Development Institute, Km4 Ondo Road, MB 611, Akure, Nigeria.

E-mail: atoyelami@yahoo.com*

ABSTRACT

This work uses two of the world's leading FEA softwares - Pro/E Mechanica and COMSOL Multiphysics - to model the structural behavior of mild steel component welded with three distinctive brands of electrode using different approaches in the two softwares. The work utilized the approach of h-elements of low order interpolating polynomials for convergence in COMSOL Multiphysics while convergence was obtained in Pro/E Mechanica by increasing the order of the interpolating polynomials on each element, that is, using p-elements. The work demonstrates the versatility of the state-of-the-art FEA software – COMSOL Multiphysics – most especially in the area of structural analysis, in comparison to a less versatile but highly resourceful in structural and thermal applications – Pro/E Mechanica. The maximum stress level (von Mises Stress) and static deflection found were compared with the material's yield strength and the time-dependent analysis respectively. The results obtained show the alarming deviation of some critical properties of mild steel and its electrodes in Nigerian markets when compared to the standards.

(Keywords: Pro-E Mechanica, COMSOL Multiphysics, FEA, h-elements, p-elements, convergence, von Mises stress, modeling and simulation)

INTRODUCTION

Mathematical modeling has become an important part of the research and development work in engineering and science[5]. Competitive edge requires speed on the path between idea and prototype, and mathematical modeling and simulation provides a valuable shortcut for understanding both qualitative and quantitative aspects of scientific and engineering design[11]. COMSOL Multiphysics offers state-of-the art performance, being built from the foundation with Java3D interface and C/C++ solvers[4]. Pro-E Mechanica or simply MECHANICA is Finite Elements Analysis software which offers more than simply being an FEA engine. It is a design tool which allows parametric studies as well as design optimization to be set up quite easily. Moreover, unlike many other commercial FEM programs where determining accuracy can be difficult or time consuming, MECHANICA is able to compute results with some certainty as to the accuracy. This refers to the issue of convergence whereby the FEA results must be verified or tested so that they can be trusted[12]. An attempt has therefore been made in this work to model and simulate the structural behavior of steel component welded with different brands of electrodes using the leading FEA software – Pro/E Mechanica and COMSOL Multiphysics.

The validity of the modeling and simulation was ascertained by comparison of the results obtained from the modeling and simulation to that obtained from experimental analysis. The essence of the work is to give an insight into an area of engineering application of the softwares and more importantly to ascertain the quality of mild steel and its electrodes in Nigerian market through the study of their structural behavior.

Progressive building collapse occurs when failure of a structural component leads to the failure and collapse of surrounding members, possibly promoting additional collapse[1]. There has been an unprecedented increase in the rate at which houses and structures are collapsing in Nigeria. The blame has always been put at the doorstep of the structural engineers who are often accused of incompetence. Findings however show that most of the times the fault is traceable to the use of sub-standard materials. The Structural Engineer for example might have based his design on the standard values of some critical properties of mild steel whereas results of this work show significant deviation of such critical properties from the established standard values.

Mild steel is the most common form of steel as its price is relatively low while it provides material properties that are acceptable for many applications. Mild steel has a low carbon content (up to 0.3%) and is therefore neither extremely brittle nor ductile[3]. It becomes malleable when heated, and so can be forged. It is also often used where large amounts of steel need to be formed, for example as structural steel. Density of this metal is 7,861.093 kg/m³, the tensile strength is a maximum of 500 MPa and it has a Young's modulus of 210 GPa[17].

SMAW (shielded metal arc welding) electrodes are covered rods manufactured by concentrically extruding silicated chemical mixtures followed by an oven cure[18]. The SMAW process uses an arc between a covered electrode and the weld pool. The arc and molten metal are shielded by the decomposition of the electrode covering. The SMAW electrode also supplies filler metal to the weld pool and to the weld metal. Ideally, the weldment produced by SMAW is expected to have approximately the same mechanical properties with the parent metals. This work is therefore an attempt to check the level of homogeneity of the weld properties when compared to the parent metals.

BASIC THEORY

The Finite Element Analysis (or Finite Element Method) is a numerical analysis technique used by Engineers, Scientists, and Mathematicians to obtain solutions to the differential equations that describe, or approximately describe a wide variety of physical (and non-physical) problems[13]. Physical problems range in diversity from solid, fluid and soil mechanics, to electromagnetism or dynamics. Finite Element Analysis uses computer simulation technique for engineering analysis. It uses a numerical technique called the finite element method (FEM)[8].

The underlying premise of the method states that a complicated domain can be sub-divided into a series of smaller regions in which the differential equations are approximately solved. By assembling the set of equations for each region, the behavior over the entire problem domain is determined. Each region is referred to as an element and the process of subdividing a domain into a finite number of elements is referred to as discretization. Elements are connected at specific points, called nodes, and the assembly process requires that the solution be continuous along common boundaries of adjacent elements.

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[5]. 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.

Processing Steps

Oftentimes, 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. In order to simplify the model, some assumptions are made. One of such assumptions 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.

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.

Convergence of H-Elements

Most programs/PDE solution software (including COMSOL Multiphysics) use this classic approach of low order interpolating polynomials in each element[4]. The interpolating functions are typically linear (first order) within each element. In order to get a more accurate estimate of the stress, it is often necessary to use much smaller elements. This process is called ***mesh refinement.*** It may not always be possible to easily identify regions where mesh refinement is required, and quite often the entire mesh is modified. The process of mesh refinement continues until further mesh division and refinement does not lead to significant changes in the obtained solution. The process of continued mesh refinement leading to an acceptable solution is called ***convergence analysis.***

Convergence of P-Elements

This convergence method is used by Pro/E Mechanica. Instead of constantly refining and recreating finer and finer meshes, convergence is obtained by *increasing the order of the interpolating polynomials on each element*. The mesh stays the same for every iteration, called a *p-loop pass.* The use of higher order interpolating polynomials for convergence analysis leads to *p-element* class of FEA methods, where the ‘p’ denotes polynomial[12]. P-elements have a different mathematical formulation when compared to h-elements:

- P-elements have shape functions with variable polynomial order. Convergence is achieved by increasing the polynomial order for a given mesh.
- H-elements have a fixed shape function polynomial. Convergence is achieved by refining the mesh in regions where higher accuracy is needed.
- P-elements can capture more complex behavior with a coarser mesh than h-elements, but are more resource intensive, both computer memory and disk space.
- With P-elements, the user is not responsible for manually designing an accurate mesh.
- In ANSYS, p-elements cannot be mixed with h-elements and p-elements must be higher-order.
- Recent advances in (Mechanica) P-element capabilities: large deformation and contact problems, transient thermal and structural behavior, 2D modeling

METHODS

Four different specimens of mild steel of configuration shown in Figure 1 were prepared. One of the specimens was used as control while each of the remaining three was cut into two (mid-way) before being joined together by one of the three brands of the popular electrodes in Nigerian market (i.e., Oerlikon, FED and China electrodes). The actual mechanical properties of each of the specimen were obtained using the Universal Testing Machine and software driven micro-hardness testing machine.

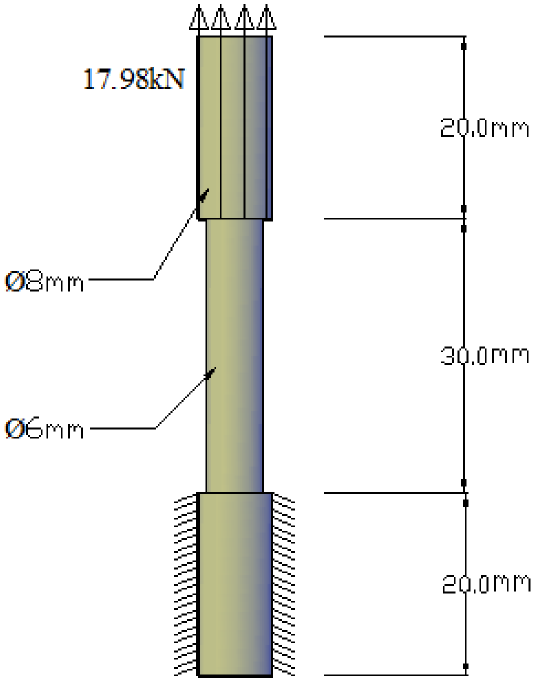

Figure 1: Mild Steel Test Piece.

The results of the testing are as shown in Figures 2 to 5 and Tables 1 and 3.

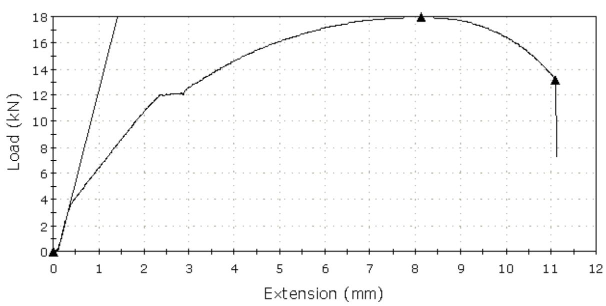

Figure 2: Graph of Load vs. Extension in the Mild Steel Control Test Piece.

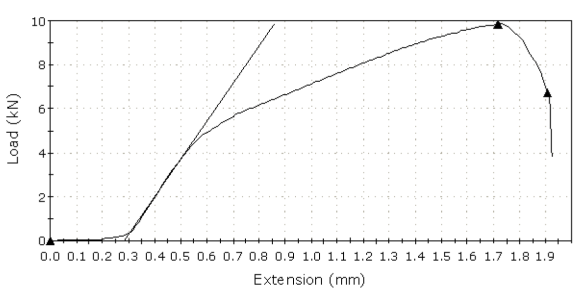

Figure 3: Graph of Load vs. Extension in the Mild Steel Test Piece welded with Oerlikon Electrode.

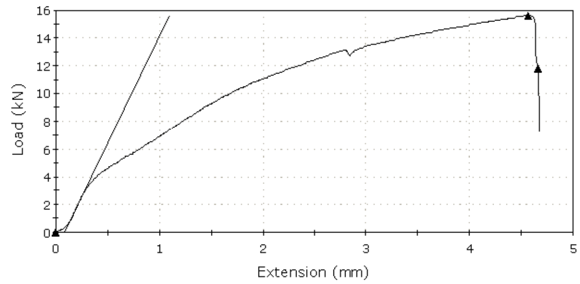

Figure 4: Graph of Load vs. Extension in the Mild Steel Test Piece welded with FED Electrode.

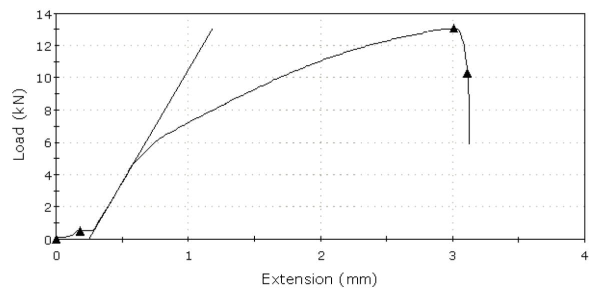

Figure 5: Graph of Load vs. Extension in the Mild Steel Test Piece welded with China Electrode.

All the results obtained were later used as input in modeling and simulating the structural behavior of steel. As illustrated in Figure 1, a longitudinal load is applied at the top end of the control test piece (this load varies depending on the experimental results obtained for a particular specimen).

Boundary conditions are then set based on the fixed and free surfaces. Since the geometry was imported from Pro/E both to the COMSOL and MECHANICA environments, a single sub-domain was used while carrying out the static analysis. A static analysis has no explicit or implicit time dependencies[4]. This situation corresponds to the steady state of a transient analysis with constant boundary conditions and material properties which is normally aimed at finding:

- The maximum stress level and compare it with the material's yield strength
- The static deflection at the point where the load is applied and compare it with the time-dependent analysis.

Table 1: Experimental Results of the Mechanical Testing of Mild Steel Test Pieces Welded with three different brands of Popular Mild Steel Electrodes in Nigerian Market.

Parameter	SPECIMEN LABEL			
	S1	S2	S3	S4
Length (mm)	27.27000	28.0400	27.7000	27.520
Diameter (mm)	5.97000	5.94000	6.04000	5.8600
Area (mm^2)	27.99230	27.7117	28.6526	26.9703
Maximum Load (kN)	17.98000	9.84000	15.6100	13.070
Tensile stress at Maximum Load (MPa)	642.2800	355.190	544.640	484.440
Tensile strain at Maximum Load (%)	29.83000	6.12000	16.4900	10.9300
Load at Break (Standard) (kN)	13.17000	6.74000	11.7800	10.3000
Extension at Break (Standard) (mm)	11.09987	1.90850	4.65906	3.11687
Tensile stress at Break (Standard) (MPa)	470.4000	243.060	411.130	381.990
Tensile strain at Break (Standard) (%)	40.70000	6.81000	16.8200	11.3300
Tensile stress at Yield (Zero Slope) (MPa)	642.2800	355.190	-	19.1500
Modulus (E-modulus) (MPa)	13205.520	17277.6	14922.3	14124.8
Energy at Break (Standard) (J)	156.75044	11.2175	49.7934	26.3278
Energy at Maximum Load (J)	107.51298	9.52872	48.4319	24.9960
Energy at Yield (Zero Slope) (J)	107.51298	9.52872	-	0.03532
Extension at Yield (Zero Slope) (mm)	8.13337	1.71669	-	0.17487
Load at Yield (Zero Slope) (N)	17978.825	9842.91	-	516.466
Tensile strain at Yield (Zero Slope) (mm/mm)	0.29825	0.06122	-	0.00635
Energy to X-Intercept at Modulus (E-modulus) (J)	0.00747	0.02069	0.01525	0.06996

Legend
S1 - Mild Steel Control Test Piece
S2 - Mild Steel Test Piece welded with Oerlikon Electrode
S3 - Mild Steel Test Piece welded with FED Electrode
S4 - Mild Steel Test Piece welded with China Electrode

RESULTS AND DISCUSSIONS

The various results of the experimental analyses and simulation and modeling are as shown in Figures 1 to 23 and Tables 1 to 4. The different approaches used for convergence in the two softwares (*h- and p-elements*) are evidenced in the density of the different meshes generated.

FEA Mesh in MECHANICA

The FEA mesh created by the automatic mesh generator is as shown in Figure 6. This FEA mesh will be taken to be a very poor mesh/ discretization by people who are used to some other FEA software which use *h-elements.* The seemingly justifiable reasons for such an allusion are:

- There are not very many elements involved in the model
- There are many long, slender elements (high aspect ratio)
- Element corners, even within the same element, can have very different angles
- Transitions in element size through the mesh are quite abrupt (long, slender elements are adjacent to short, wide ones)

In an h-code based method, all these would be signs of a poorly constructed and possibly lethal mesh, and would raise serious concerns about the accuracy of the solution.

In order to allay such palpable fear, some of the limits in the Mesh Generator default setting (AutoGem) was tampered with as shown in Figure 7. This adjustment resulted into increase in the number of the meshing elements from 170 to 2,251 (Figures 6 and 7).

As can be seen from Table 2, there are no significant differences between the Von Mises Stresses (1.53MPa and 1.48MPa) and the maximum displacement values (0.12213mm and 0.12203mm) whereas the solution time significantly increases from 20.30s to 153.50s and the system memory usage increases from about 203MB to about 1.6GB. This shows that the mesh density in MECHANICA does not have a significant effect on the results in the final solution, although it significantly affects the solution time and the memory usage.

Table 2: Model Summary for Design Study on "Steel Analysis" using Pro Mechanica

S/N	NAME	Default Meshing		Refined Meshing	
		Value	Convergence (%)	Value	Convergence (%)
1	Maximum Displacement Magnitude (mm)	1.22129e-01	0.1	1.22028e-01	0.0
2	Maximum X Displacement over Model (mm)	-3.0504e-03	0.2	2.58718e-03	0.0
3	Maximum Y Displacement over Model (mm)	1.22108e-01	0.1	1.22022e-01	0.0
4	Maximum Z Displacement over Model (mm)	2.87922e-03	5.0	2.75565e-03	4.1
5	Maximum Von Mises Stress (N/mm^2)	1.5307e+03	2.7	1.4851e+03	12.7
6	Maximum XX Stress Component (N/mm^2)	6.3655e+02	12.1	1.1555e+03	18.8
7	Maximum XY Stress Component (N/mm^2)	4.3221e+02	0.6	7.1483e+02	15.9
8	Maximum XZ Stress Component (N/mm^2)	1.9253e+02	11.3	-2.0816e+02	8.6
9	Maximum YY Stress Component (N/mm^2)	1.6322e+03	5.4	1.7433e+03	14.5
10	Maximum YZ Stress Component (N/mm^2)	-6.5061e+02	0.7	6.1835e+02	15.5
11	Maximum ZZ Stress Component (N/mm^2)	7.0775e+02	0.4	9.3754e+02	18.3
12	Points	72		618	
13	Edges	302		3265	
14	Faces	401		4899	
15	Elements	170		2251	
16	Total Elapsed Time (s)	19.96		144.93	
17	Total CPU Time (s)	20.30		153.50	
18	Maximum Memory Usage (KB)	203375		1627923	
19	Working Directory Disk Usage (KB)	47104		118784	

Table 3: Hardness Values Obtained using Micro Hardness Tester.

Sample	HV
China electrode weldment	260.5
FED electrode Weldment	244.8
Oerlikon electrode weldment	204.1

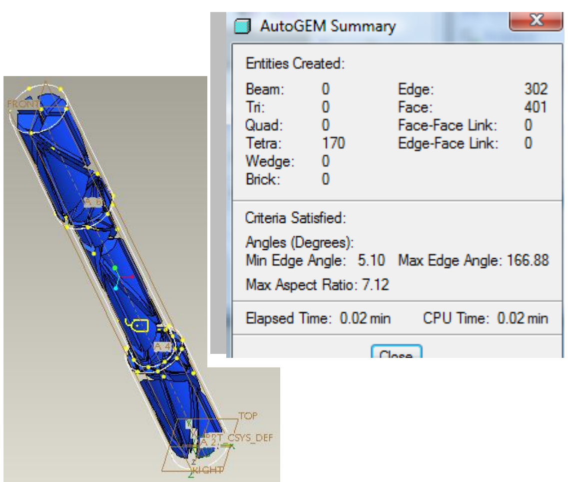

Figure 6: Mesh Generated by MECHANICA using Default Setting (consisting of 170 elements).

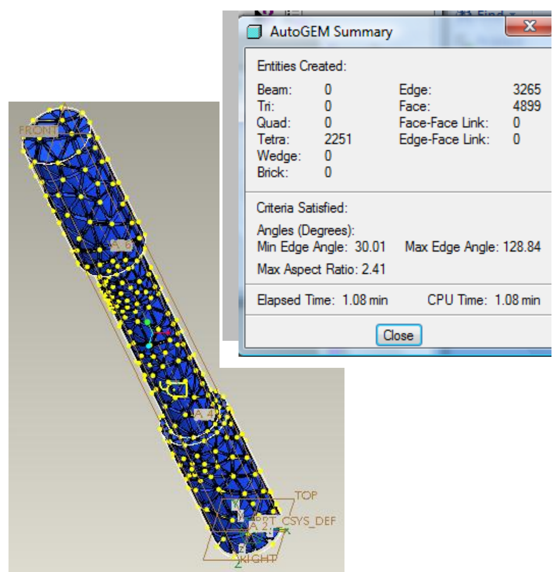

Figure 7: Mesh Generated by MECHANICA using User Defined Setting (consisting of 2251 elements).

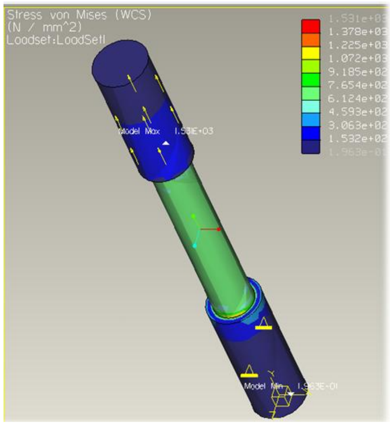

Figure 8: Von Mises Stress Distribution using MECHANICA.

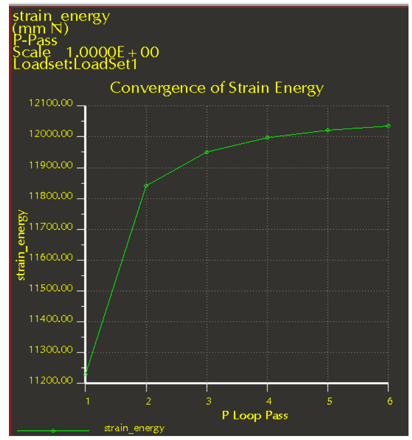

Figure 11: Convergence Test in MECHANICA using Strain Energy.

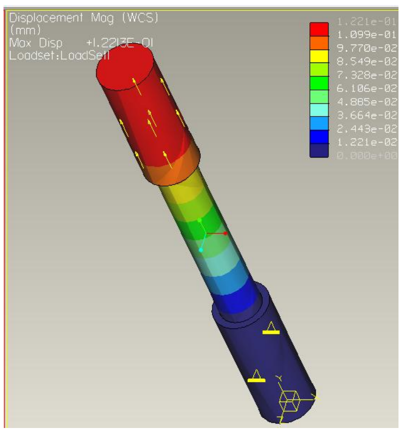

Figure 9: Displacement/Extension before Breaking using MECHANICA.

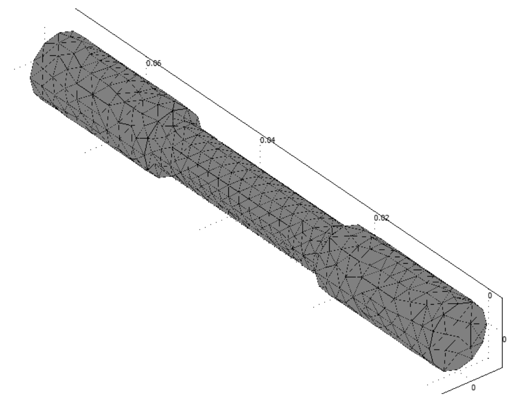

Figure 12: Initial Mesh Generated using COMSOL Multiphysics (consisting of 3115 elements).

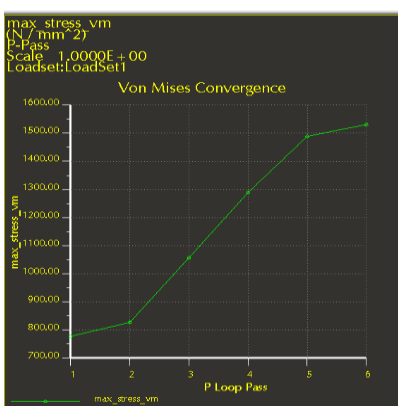

Figure 10: Convergence Test in MECHANICA using Von Mises Stress.

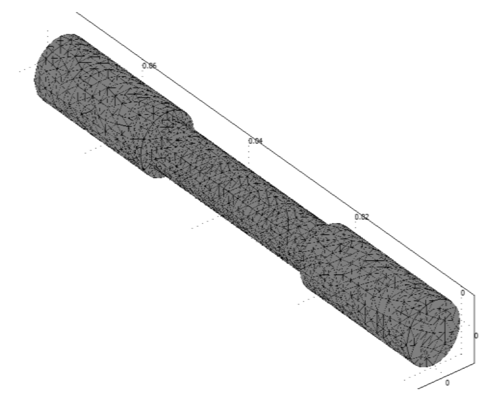

Figure 13: First Mesh Refinement using COMSOL Multiphysics (consisting of 11167 elements).

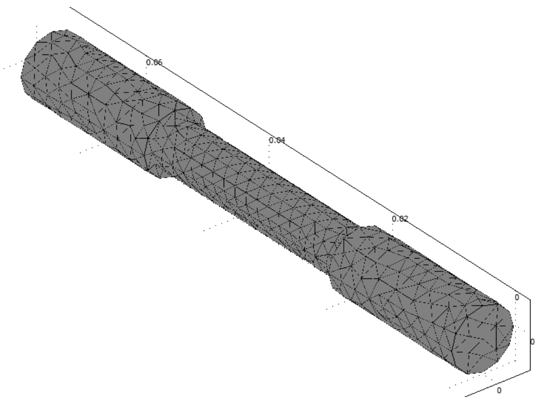

Figure 14: Second Mesh Refinement using COMSOL Multiphysics (consisting of 34525 elements).

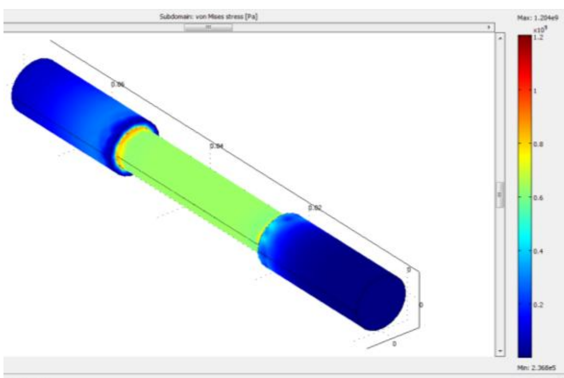

Figure 15: Solution Obtained for the Sub-domain Von Mises Stress using Initial Mesh Generated in COMSOL in Pa (Degrees of freedom = 16,656; Solution time = 2.127s).

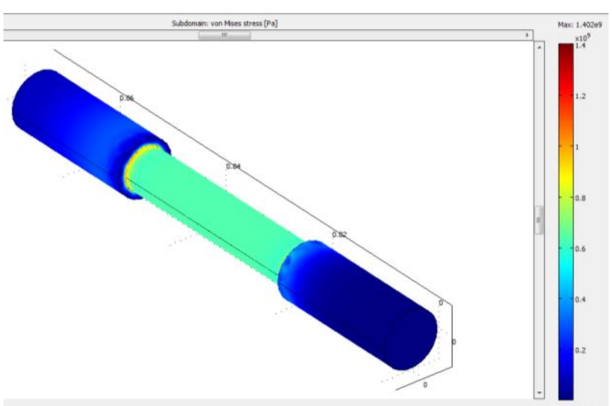

Figure 16: Solution Obtained for the Sub-domain Von Mises Stress using First Mesh Refinement in COMSOL [in Pa] (Degrees of freedom = 51,696; Solution time = 8.394s).

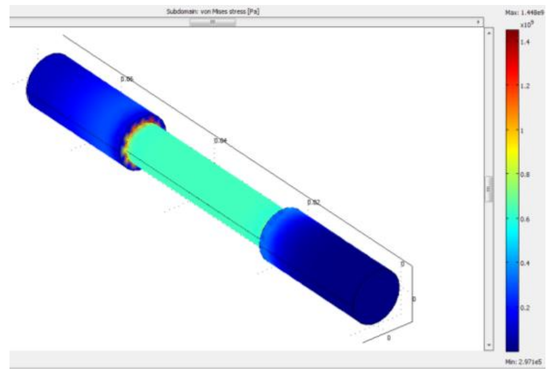

Figure 17: Solution Obtained for the Sub-domain Von Mises Stress using Second Mesh Refinement in COMSOL [in Pa] (Degrees of freedom = 154,050; Solution time = 80.811s)

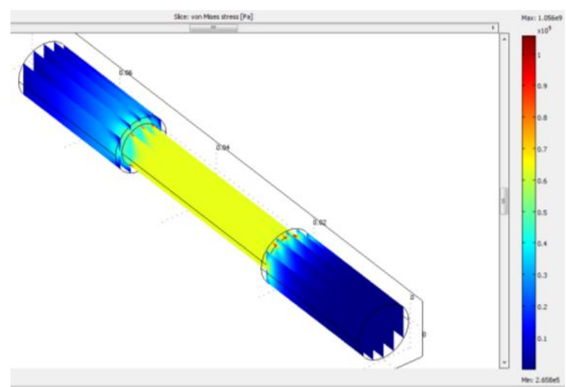

Figure 18: Solution Obtained for the Slice Von Mises Stress using Second Mesh Refinement in COMSOL [in Pa].

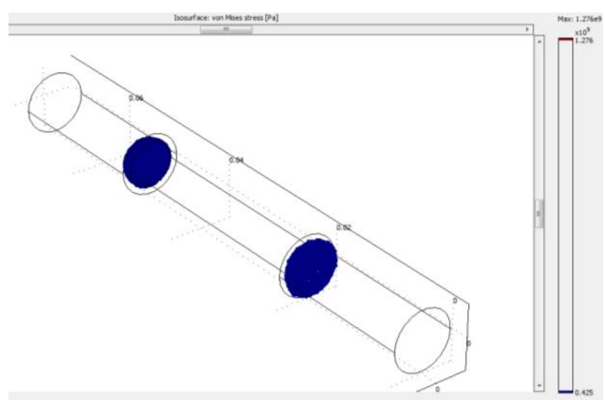

Figure 19: Solution Obtained for the Isosurface Von Mises Stress using Second Mesh Refinement in COMSOL [in Pa].

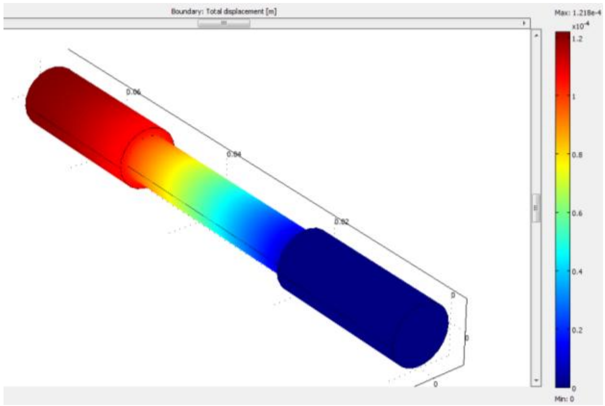

Figure 20: Solution Obtained for the Boundary Total Displacement using Second Mesh Refinement in COMSOL [in m].

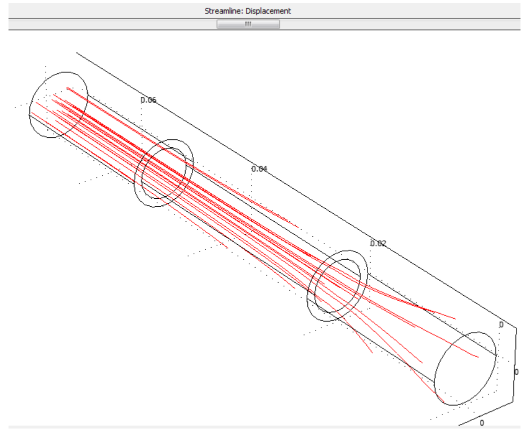

Figure 23: Solution Obtained for the Streamline Displacement using Second Mesh Refinement in COMSOL.

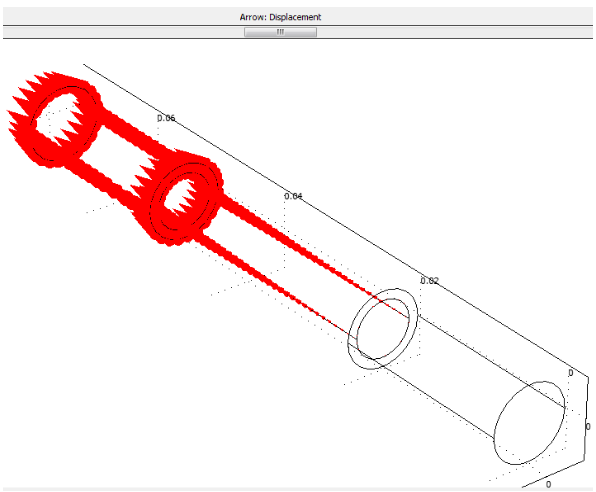

Figure 21: Solution Obtained for the Arrow Displacement using Second Mesh Refinement in COMSOL.

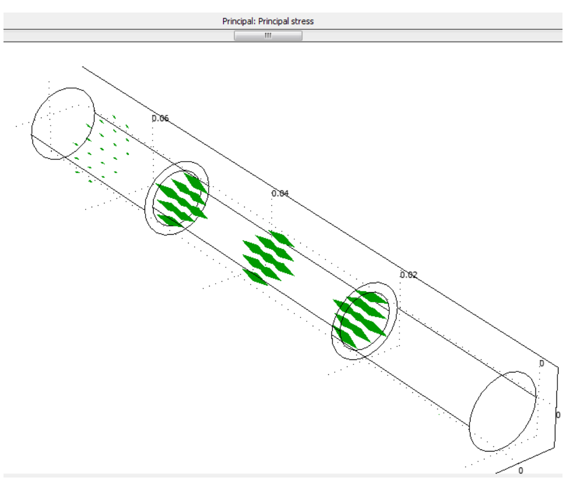

Figure 22: Solution Obtained for the Principal Stress using Second Mesh Refinement in COMSOL.

Table 4: Summarized Results of the Modeling and Simulation of Steel Welded with Different Brands of Electrode using Pro Mechanica and COMSOL Multiphysics.

Test Piece	ST	S1	S2	S3	S4
Actual Young Modulus (MPa)	199948	13205.52	17277.61	14922.29	14124.77
Actual Deformation (mm)		8.13337	1.71669	-	0.17487
Actual Tensile Stress at Yield (MPa)		642.28	355.19	-	19.15
Displacement using MECHANICA (mm)	0.1221	1.849	1.413	1.072	0.9904
Displacement using COMSOL (mm)	0.1218	0.1845	0.7716	0.1417	1.254
Von Mises Stress using MECHANICA (MPa)	612.4	612.4	612.4	401.3	350.8
Von Mises Stress using COMSOL (MPa)	610	610	350	500	420

Legend

ST - Standard Steel
S1 - Mild Steel Control Test Piece
S2 - Mild Steel Test Piece welded with Oerlikon Electrode
S3 - Mild Steel Test Piece welded with FED Electrode
S4 - Mild Steel Test Piece welded with China Electrode

FEA Mesh in COMSOL

The default h-element type in COMSOL Multiphysics is quadratic Lagrange elements. The concept uses 2nd-order polynomials, which is often a good trade-off between memory usage and accuracy. Linear Lagrange elements are used to reduce memory consumption when accurate stress or strain results are not required.

To improve the level of accuracy, the mesh is refined and the analysis is run again. It is then observed if the solution is converging to a stable value as the mesh is refined. If the solution changes when the mesh is refined, the solution is mesh dependent, so the model requires a finer mesh. This approach was used to generate the mesh until it was established that there was no significant difference between the current results and the preceding ones. All the meshes generated and the solutions obtained are as shown in Figures 12 to 23 and Table 4.

CONCLUSION

Structural behavior of mild steel component welded with three most popular mild steel electrodes in Nigerian market (Oerlikon, FED and China electrodes) has been investigated. Results of the experimental analysis carried out through mechanical testing of the components were used to model and simulate several other structural properties of mild steel components using-state-of-the-art FEA software – Pro/E Mechanica and COMSOL Multiphysics. The various results of the experimental analyses and that of the modeling and simulation are as shown in Figures 1 to 23 and Tables 1 to 4.

Some of the critical properties of mild steel obtained from the experimental analysis carried out vary significantly from the known standard values. The most prominent of them are the Young Modulus and Yield Strength of Steel. The Young Modulus is supposed to be between 190 and 210MPa [6, 7] while the Yield Strength is 353.4 [16]. The values obtained from the experimental analysis are however at a great variance from the standard values. The summarized results of the experimental analysis together with that of the Modeling and Simulation using Pro Mechanica and COMSOL Multiphysics are as shown in Table 4.

REFERENCES

1. Baldridge, S.M. and Humay, F.K. 2003. "Preventing Progressive Collapse in Concrete Buildings". *Concrete International*. November 2003:73-79.

2. BOC Gases New Zealand Limited. 2001. "General Purpose Mild Steel Electrodes: Statement of Hazardous Nature". BOC Gases: Auckland, NZ.

3. Carvill, J. 2003. *Mechanical Engineer's Data Handbook*. Butterworth-Heinemann Jordan Hill: Burlington MA.

4. COMSOL. 2008. *Multiphysics Modeling Guide, Version 3.4*. COMSOL: Burlington, MA.

5. Crisfield, M.A. 2000. *Non-Linear Finite Element Analysis of Solids and Structures, Volume 1.* John Wiley & Sons: West Sussex, England.

6. Dan, B.M. 2001. *Mechanical Engineer's Handbook*. Academic Press: San Diego, CA.

7. Edward, H.S. 2000. *Mechanical Engineer's Reference Book*. Buttenvorth-Heinemann Linacre House, Jordan Hill: Woburn, MA.

8. Friedel, H. and Casimir, K. 2007. *Structural Analysis with Finite Elements*. Springer-Verlag: Berlin, Germany.

9. Hartsuijker, C. and Welleman, J.W. 2006. *Engineering Mechanics*. Springer: Dordrecht, The Netherlands.

10. UNM. 2006. *Manual on Tensile Test of Steel*. University of New Mexico, Civil Engineering Department, Civil Engineering Materials Laboratory, CE 305L.

11. Pandey, M., Wei-Chau, X., and Lei, X. 2006. "Advances in Engineering Structures, Mechanics & Construction". *Proceedings of an International Conference on Advances in Engineering Structures, Mechanics & Construction*. Waterloo, Ontario, Canada, May 14-17.

12. Pro/ENGINEER® Wildfire™ 3.0. 2006. *Structural and Thermal Simulation Help Topic Collection* Parametric Technology Corporation: Needham, MA.

13. Tapan, S., Daniel, L., and Theodor, K. 2005. "Finite Element Analysis of Steel Beam to Column Connections Subjected to Blast Loads". *International Journal of Impact Engineering*. 31: 861–876.

14. http://www.efunda.com/materials/alloys/carbon_steels/show_carbon.cfm?ID=AISI_1040&prop=all&Page_Title=AISI%2010xx

15. http://www.springerlink.com/content/q550jk/

16. http://www.efunda.com/

17. file:///C:/w/index.php?title=Carbon_steel&action=edit&section=1

18. http://www.wipo.int/portal/index.html.en

SUGGESTED CITATION

Oyelami, A.T., A.K. Ogunkoya, O.D. Ogundare, and B.A. Olunlade. 2012. "Finite Elements Analysis of Weldments with H- and P-Elements of Differing Orders of Interpolating Polynomials". *Pacific Journal of Science and Technology*. 13(1):67-77.

Pacific Journal of Science and Technology