IJRAME

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.6 Issue.7,
July 2018
Pg: -01-07

"Finite Element Analysis of Weld Joint in Shaft and validation Using Experimental Method"

1st ANURAG MISHRA[1], 2nd Mr. PRADEEP SAHU[2]

[1] M Tech Research Scholar SSIPMT Raipur

[2] Assistant Professor, Mechanical Engineering SSIPMT, RAIPUR

[1]1st –anuragmishraji810@gmail.com, [2]2nd –pradeepsahu231@gmail.com

Abstract— ***In this research work welding simulation was carries out by experimental and also on FEA software to find out the value of stress and deflection under different loading condition. The numerical simulations show that the concentration of stress is maximum near to the joint and at the corner where the cross section has suddenly changed. In order to find out the most optimum load, the joint can bear is calculated by using experimental method in modern universal testing machine. It is very important to check & compare the FEA simulation result with the result of physical test or experiment in which the real behavior of specimen can observed. The research work deal with the physical test of specimen whose main purpose in to find out the optimum material, geometry and strength characteristics of butt weld joint. The specimen of weld joint is tested on UTM machine and corresponding output parameter as stress and deflection is monitored. The result obtained by physical experiment is validated by using different numerical approaches as Solid works and ANSYS simulation.***

Keywords— FEA Finite element analysis, FVM- Finite Volume Method, 2 D two dimensional, FDM- Finite Difference Method, UTM- Universal Testing Machine

I. INTRODUCTION

With the expansion of interest for both high generation rates and high exactness, completely motorized or mechanized welding forms have assumed a conspicuous position in the welding field. The rate at which mechanization is being brought into welding process is surprising and it might be normal that before this present century's over more computerized machines than men in welding manufacture units will be found. The PCs assume imperative part in running the computerized welding forms and the direction given by the PC will be taken from the projects, which thusly, require calculations of the welding factors as mathematical conditions. To make powerful utilization of the robotized frameworks it is basic that a high level of certainty be accomplished in anticipating the weld parameters to achieve the coveted mechanical quality in welded joints. To create scientific models to precisely foresee the weld quality to be nourished to the robotized welding frameworks has turned out to be more basic.

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.6 Issue.7,
July 2018
Pg: -01-07

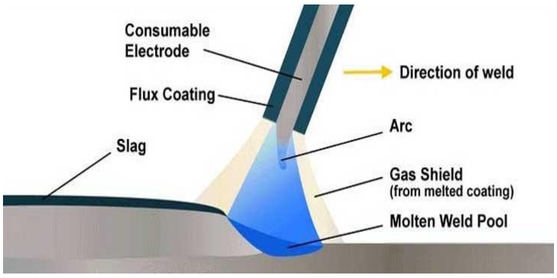

Figure 1 welding process

Residual Stresses

Leftover burdens are stresses that would exist in a body if every outside load were evacuated. They are some of the time called interior anxieties. Remaining anxieties that exist in a body that has already been subjected to non-uniform temperature changes, for example, those amid welding, are regularly called warm burdens. Generally, the weld metal and the nearby base metal are similar to the centre bar, and the territories more remote far from the weld metal closely resemble the two sidebars. This is because the development and compression of the weld metal and the nearby base metal are limited by the regions more distant far from the weld metal.

II. PROBLEM IDENTIFICATION

The present study deals with finite element analysis of welded butt joint of circular bar of 13 mm diameter under the different tensile loading of alloy steel. In this research work, welding strength of material is calculated by using FEA technique. In order to find out the behavior of stress distribution and plot of deflection is verified by applying the experimental method with UTM machine.

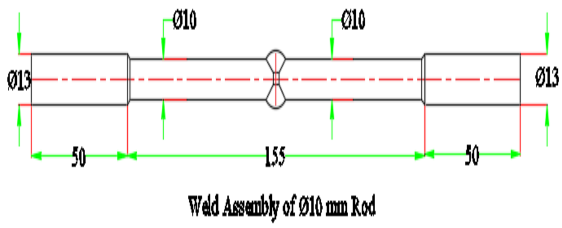

Fig. 2 Schematic representation of welded joint

An edge preparation has done before performing the welding process for proper welding process. Same boundary condition has applied for analysis of weld strength.

Fig. 3 mild steel specimen of 10 mm diameter

III. METHODOLOGY

1. Experimental Method:

A specimen were machined and welded to investigate the stress in butt joint. Specimen designed to show the detail that the strength of connected element has to be higher than the strength of the welded jointThe rod is turned into lathe to get the desire shape and size and after that, it cut in heck saw and prepared edge for proper welding process. After the edge preparation, the specimen is welded by using arc-welding process with 6 mm weld size.

A universal testing machine having specimen holding unit to assemble the job in proper orientation, generally for tensile test it's available in upper end of the machine. the object or specimen are fixed with the specimen holding tools and gradually applying the load to know the nature of deflection of specimen and its stress distribution with respect to the load. A control unit is also available with the modern universal testing machine to plot and print the graph of stress-strain curve of object.

The welded rod consist of 10 mm diameter is tested on UTM machine and start with the load 6 KN and the graph plotted with the increment of 6 KN, corresponding deflection is recorded and plotted on the graph.

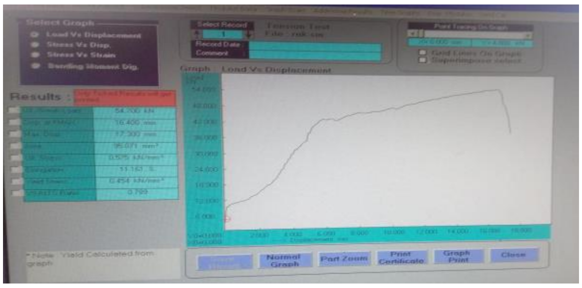

Fig.4 Distribution of deflection of dia. 10 mm

2. FEA analysis of welding joints

(a) Modeling of the Geometry

The modeling of actual model of component is very complex problem in FEA so a proper care and attention is required to create the geometry of component. Ansys and solid works provide a separate workbench to create the desire shape and size of model as per the requirement. The curves part is modeled through the complex geometry tool like sleeve and loft. However, it should be noted that the geometry is equally divided into number of straight line or plane to mesh proper size for further analysis.

IJRAME

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.6 Issue.7,
July 2018
Pg: -01-07

(b) Meshing

Meshing is performed to discretize the geometry created into small pieces called elements or cells. Lattice is performed to discretize the geometry made into little pieces called components or cells. The balanced behind lattice can be clarified in an extremely clear and consistent way. We can expect the answer for a building issue to be extremely perplexing, and fluctuates in a way that is exceptionally eccentric utilizing capacities over the entire space of the issue.

(c) Boundary, Initial and Loading Conditions

Boundary, initial and loading conditions play an important role in solving the simulation. Inputting these conditions is usually done easily using commercial pre-processors, and it is often interfaced with graphics. Users can specify these conditions either to the geometrical identities (points, lines or curves, surfaces, and solids) or to the elements or grids. Again, to accurately simulate these conditions for actual engineering systems requires experience, knowledge and proper engineering judgments. The boundary, initial and loading conditions are different from problem to problem.

Numerical Solution by Solid works Simulation:

On applying the relevant boundary conditions in the form of load and constraints, we achieve the stress zones and plot of deflection. In order to clearly differentiate all three-test specimen at same boundary condition, a separate analysis file is used to investigate the nature of stress and deflection of welded rod assembly.

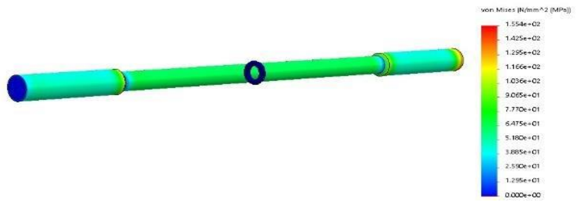

Fig.5 Stress Distribution in Butt Welded Specimen in solid works

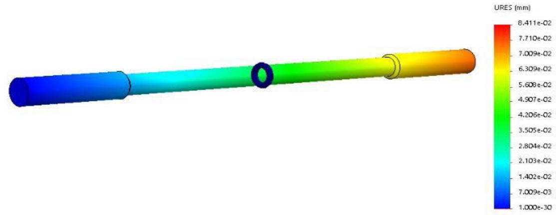

Fig.6 Deflection Plot for Butt Welded Specimen

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.6 Issue.7,
July 2018
Pg: -01-07

Solution by ANSYS Simulation:

The distribution of stress and deflection is given in the figure below. For the simulation of butt joint cylindrical rod mild steel of grade AISI 1020 material is taken for analysis.

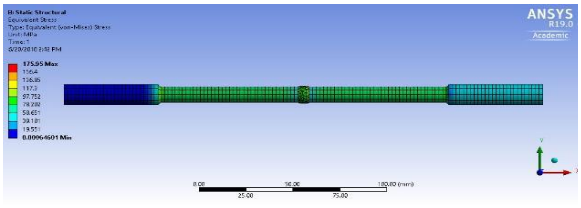

Fig.7 Stress Distribution in Butt Welded Specimen in ANSYS

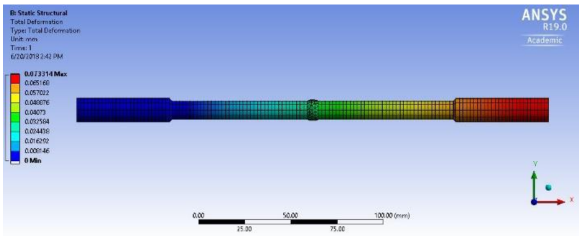

Fig. 8 Deflection Plot for Butt Welded Specimen in ANSYS

IV. RESULTS

Comparison of FEA result with Experimental method:

Stress is being calculated for the butt welded joint under tensile loading at various loading condition. The specimen is tested for different loading values from 6KN to 54KN &corresponding induced stress and deflection are calculated.

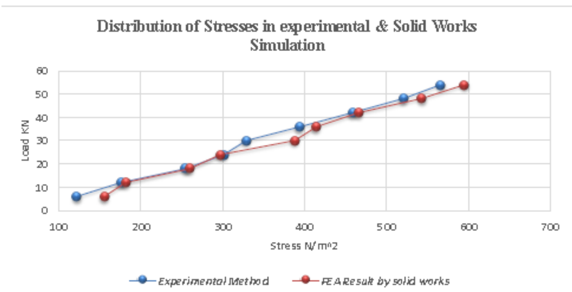

Fig. 9 Stress plot for experimental and solid works Simulation

Comparison of all Approaches

The analysis result obtained by all approaches is shown in figure below in figure 10 & 11. it clearly shows that the stress distribution shows vary good agreement in all approaches but in case of deflection plot the Ansys simulation is near to the experimental method as compare to the solid works simulation.

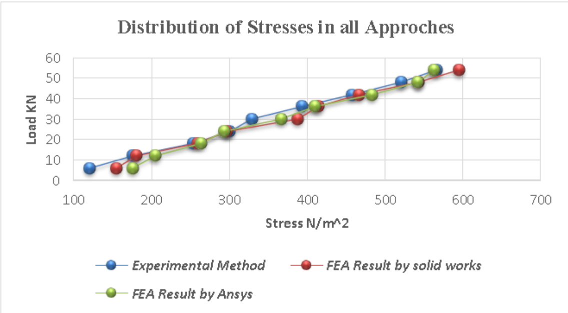

Fig. 10 Stress plot for all approaches

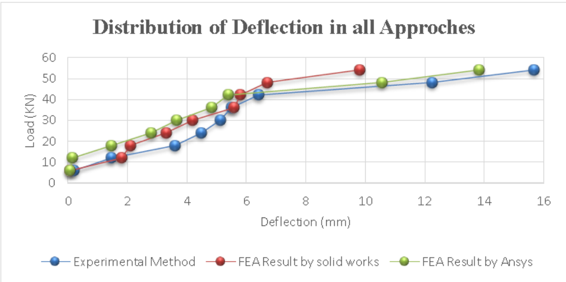

Fig. 11 Deflection plot for all approaches

V. CONCLUSIONS

- The result of study shows that the numerical method can also describe the stress and deflection state with sufficient precision.
- In case of ANSYS Simulation, we can say that this approaches is very near to the actual experimental method.

- On the basis of these experiment there were obtained parameter of real stress distribution that will be used as reference parameter for further study.
- The numerical result may more filtered by using fine meshing of cad geometry but subjected to more ram and graphics requirement which is possible only in works station.

References

[1] Martinkrejsa et al. "Numerical modeling of fillet and butt welds in steel structure elements with verification using experiment". Elsevier 190 (2017) 318-325.
[2]Andrea Capriccioli& Paolo Frosi, "Multipurpose ANSYS FE procedure for welding processes simulation", Elsevier 190 (2009) 546-553.
[3] C.M. Chen & R. Kovacevic "Finite element modeling of friction stir welding thermal and thermo mechanical analysis". Elsevier International Journal of Machine Tools & Manufacture 43 (2003) 1319–1326.
[4] ZhigangHou "Finite element analysis for the mechanical features of resistance spot welding process". Elsevier Journal of Materials Processing Technology 185 (2007) 160–165.
[5] Fenggui Lu et al. "Modeling and finite element analysis on GTAW arc and weld pool". Elsevier Computational Materials Science 29 (2004) 371–378.
[6] Ranjit Shrestha et al,. "Numerical Simulation of Similar and Dissimilar Materials Welding Process; Quantifications of Temperature, Stress, Strain and Deformation" International Journal of Engineering and Technology, Vol 7 No 1 Feb-Mar 2015.
[7] N.B.Landg et al, "Tensile Strength Analysis of V Groove Butt Weld Joint For Aluminium Alloys Aa 2025 & Aa 7025", IJARIIE-ISSN(O)-2395-4396, Vol-2 Issue-6 2016.
[8] Vinod M. Bansodeet al, "Fatigue Life Estimation of a Butt Welded Joint by S-N Approach", International Journal of Applied Research in Mechanical Engineering, ISSN: 2231 –5950, Vol-2, Iss-2, 2012.
[9] K. A. Mohammada, et al, " Experimental Determination of the Fatigue Behavior of Austenitic 316L Stainless Steel under Fatigue and Creep-Fatigue Tests at High Temperature" International Journal of Metal and Steel Research Technology, Vol. 1, No. 1, July 2013, PP: 01- 11.
[10] Shrestha Ranjit et al, "Fatigue Analysis of Dissimilar Materials Welded Specimen Using Finite Element Analysis", International Journal of Applied Engineering Research ISSN 0973-4562 Volume 11, Number 5 (2016) pp 3390-3393.
[11] Nelson Arzola et al, "Prediction of Fatigue Life for a Transverse Fillet Welded Joint and Analysis of the Influence of Crack Eccentricity on the Failure". Dyne, year 80, No. 182, pp. 95-104. Medellin, December, 2013. ISSN 0012-7353.
[12] Thirugnanam, "Analysis of Stress in Welded Joint in Bending and in Torsion Using Ansys". Middle-East Journal of Scientific Research ISSN 1990-9233 20 (5): 580-585, 2014.