Finite Element Method of Welding Joint in Shaft and Validation Using Different Method

Anurag Mishra[1], Mr. Pradeep Sahu[2]

[1]M Tech Research Scholar SSIPMT Raipur
[2]Assistant Professor, Mechanical Engineering SSIPMT

Abstract:In this research work welding simulation was carries out by experimental and also on Solid work 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(UTM).it is very important to check & compare the solid work 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 simulation.

KEYWORDS: FEA Finite element analysis, FVM- Finite Volume Method, 2 D two dimensional, FDM- Finite Difference Method, UTM- Universal Testing Machine & Simulation of solid work.

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.

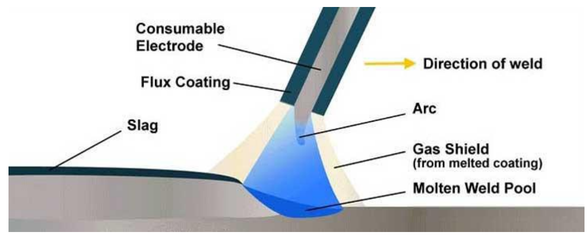

Figure 1 welding process

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.

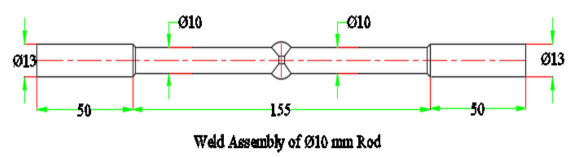

Fig. 2 Schematic representation of welded joint

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 universal testing machine

Table 4.1 Experimental Result of Butt Weld Joint dia. 10 mm

S No.	Load (KN)	Stress (N/M^2)	Deflection (mm)
1	6	121	0.2
2	12	176	1.46
3	18	254	3.6
4	24	301	4.5
5	30	329	5.11
6	36	394	5.5
7	42	458	6.4
8	48	521	12.27
9	54	566	15.7

Different type of load in specimen material and plot tha graph in utm machine.

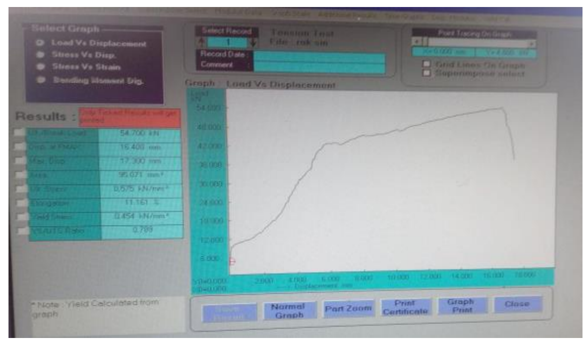

Fig.4.2 Distribution of deflection of dia. 10 mm

2. FEA analysis of welding joints

The new generation of CAD/ CAE software's followed the rapid progress in industry, focusing on manufactory. Lately, highly developed and complex features were introduced to empower CAD software, allowing it to satisfy the advanced requirements dictated by the new industry. ANSYS, CATIA, Pro-engineer, Autodesk, and Inventor are the most commonly used software's in this field. ANSYS is a powerful software tool to determine the effect of real time simulation of physical condition. It consist of a wide range of workbench to determine the desire output in the form of graphical representation of chart and curves.

The Finite Element Method (FEM) has formed into a key, imperative innovation in the displaying and reproduction of innovative designing frameworks in different fields like development transportation, interchanges, et cetera. In building such propelled designing frameworks, architects and creators experience an advanced procedure of displaying, reenactment, representation, investigation, outlining, prototyping, testing, and ultimately, manufacture. Note that much work is included before the creation of the last item or framework. This is to guarantee the usefulness of the completed item, and additionally for cost viability. The procedure is delineated as a flowchart in Fig. 4.4. This procedure is frequently iterative in nature, implying that a portion of the methodology are rehashed in view of the outcomes got at a present stage, in order to accomplish an ideal execution at the most reduced cost for the framework to be constructed. Along these lines, systems identified with displaying and reproduction in a quick and powerful way assume an inexorably essential part, bringing about the utilization of the FEM being duplicated various circumstances along these lines.

A ceaseless capacity of an obscure field variable is approximated utilizing piecewise straight capacities in each sub-space, called a component shaped by hubs. The questions are then the discrete estimations of the field variable at the hubs. Next, legitimate standards are taken after to set up conditions for the components, after which the components are 'tied' to each other. This procedure prompts an arrangement of direct arithmetical synchronous conditions for the whole framework that can be unraveled effortlessly to yield the required field variable.

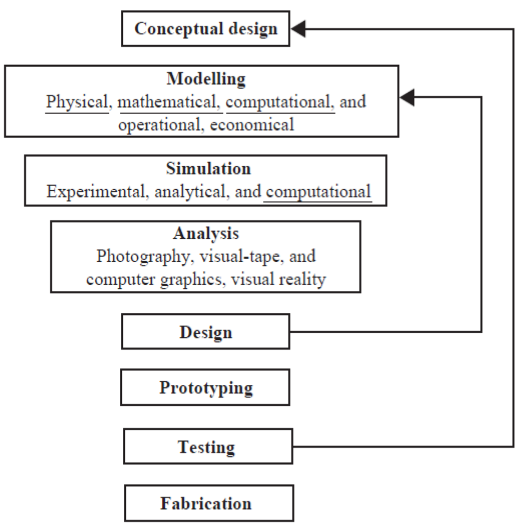

Fig. 5.1 Processes leading to fabrication of advanced engineering system

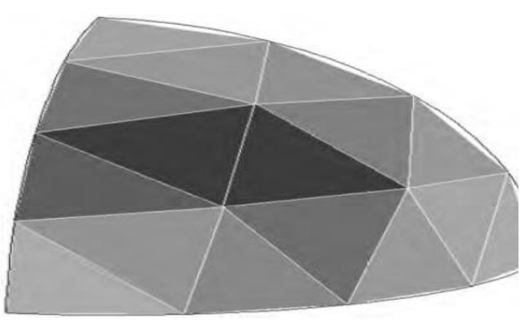

Fig. 5.2 Hemispherical section discretized into several shell elements

3. Physical Problems in Engineering

There are various physical building issues in a specific framework. As specified before, in spite of the fact that the FEM was at first utilized for pressure examination, numerous other physical issues can be tackled utilizing the FEM. Scientific models of the FEM have been figured for the numerous physical marvels in building frameworks. Regular physical issues unraveled utilizing the standard FEM include: Mechanics for solids and structures.

- Heat transfer.
- Acoustics.
- Fluid mechanics.

The conceptual understanding of the methodology of the FEM is the most important, as the application of the FEM to all other physical problems utilizes similar concepts. Computer modeling using the FEM consists of the major steps discussed in the next section.

4. Computational Modeling Using the FEM The conduct of a wonder in a framework relies on the geometry or space of the framework, the property of the material or medium, and the limit, starting and stacking conditions. For a designing framework, the geometry or space can be extremely unpredictable. Further, the limit and introductory conditions can likewise be muddled. It is accordingly, as a rule, exceptionally hard to unravel the administering differential condition through diagnostic means. Practically speaking, a large portion of the issues is fathomed utilizing numerical techniques. Among these, the techniques for area discretization championed by the FEM are the most prevalent, because of its common sense and flexibility. The system of computational displaying utilizing the FEM extensively comprises of four stages:

- Modeling of the geometry.
- Meshing (discretization).
- Specification of material property.
- Specification of boundary, initial and loading conditions.

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.

Fig. 6 CAD model of weld joint

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.

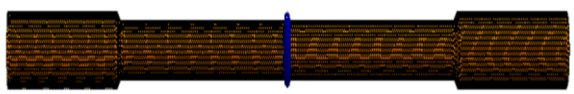

Fig. 7 Meshed model of welded assembly

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.

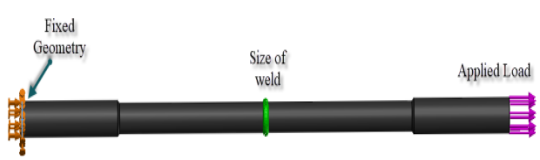

Fig. 8 Boundary condition of welded rod

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 two-test specimen at same boundary condition, a separate analysis file is used to investigate the nature of stress and deflection of welded rod assembly.

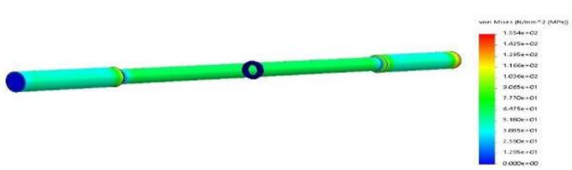

Fig.9 Stress Distribution in Butt Welded Specimen in solid works

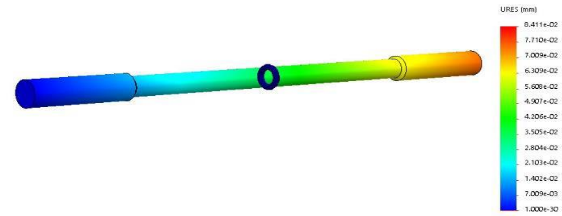

Fig.10 Stress Deflection Plot for Butt Welded Specimen

I. Results & Discussions

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.

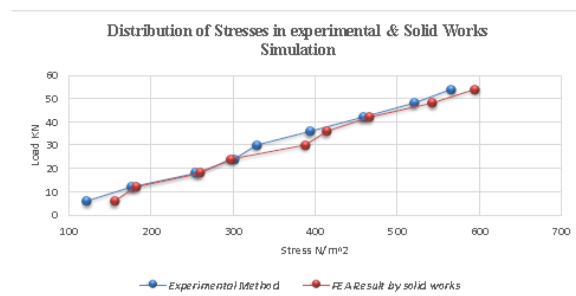

Fig. 11 Stress plot for experimental and solid works Simulation

II. Conclusion

- 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 approach is very near to the actual experimental method.
- Based onthe 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.

III.REFERENCES

[1] Martin krejsa 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 Aluminum 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". Dyna, year 80, Nro. 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.

[13] Ahmet H. Ertas And Fazil O. Sonmez "A parametric study on fatigue strength of spot-weld joints" Blackwell Publishing Ltd. Fatigue FractEngng Mater Struct 31, 766–776, 2008.

[14] DragiStamenković& Ivana "Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates" Scientific Technical Review,Vol.LIX,No.1,2009.

[15] Chi-Liang Kung et al," "Residual Stress and Deformation Analysis in Butt Welding on 6 mm SUS304 Steel with Jig Constraints Using Gas Metal Arc Welding" Appl. Sci. 2017, 7, 982; doi:10.3390/app7100982.

[16] Collins EruogunEtin-osa," Analysis of Optimum Butt Welded Joint for Mild Steel Components Using FEM (ANSYS) 2017.

AUTHOR:

ANURAG MISHRA

M.tech student in Dept.of
Mechanical engineering ,SSIPMT
Raipur
Chhattisgarh ,India.