International Journal of Interdisciplinary Innovative Research &Development (IJIIRD)
ISSN: 2456-236X
Vol. **05** Special Issue **01 | 2020**

Finite Element Modeling Of Welded Pipe Joint For Bending And Tensile Stresses

Mr.Pranav wagh, Dr. Atul Kulkarni, Mr. Piyush Rode
[1] *Mr.Pranav wagh , Mechanical Engineering Department, VIIT Pune ,Maharashtra ,India*
[2] *Dr. Atul Kulkarni Mechanical Engineering Department, VIIT Pune, Maharashtra ,India*
[3] *Mr. Piyush Rode , Dy.Manager Bharat forge, Mundhwa Pune, Maharashtra ,India*

ABSTRACT

This article presents the finite element modelling of welded pipe joint with v-chamfer penetration. static structural analysis is performed on the welded pipe joint of two deferent materials. Material of pipe structural steel and welding rod material is CB08Г2C.the analysis takes into consideration equivalent stresses induced in welded joint, total deformation in welded joint and Equivalent elastic strain. Material properties, modelling procedure, loading conditions to perform static structural analysis is explained in the article. this welded joint has automobile and heavy engineering applications. .

Keyword : - welding, static analysis, equivalent stress, equivalent elastic strain ,bending stress, tensile stress

1.Introduction

Anticipating the equivalent stresses and distortions is very important for welded structures. This study is a part of larger project in which attempts are made to find out stresses induced in the welded joint of structural steel and CB08Г2C which has application in heavy automobile machinery

Our goal is to build reliable finite element model for static structural analysis of welded pipe joint to find out equivalent stresses induced in welded joint .

2.Literature survey

Ramesh Sawant, Sachin Awasare and Asokendu Samanta in their paper titled "Finite element analysis of welded structures**"** have studied that in welding process, there are many parameters, which affect the distortion and residual stress distribution in a welded plate. And made attempts to determine temperature distribution due to welding by finite element method in welded panel structures. They used ANSYS for this purpose. We get the idea of A nonlinear transient thermal analysis from this paper . [1]

A. Krasovskyy*, S. Sönnichsen, D. Bachmann in their paper titled "On the residual stresses in multi-pass welds: coupling of welding simulation and fatigue analysis" studied fatigue assessment for multi-pass welds based on the simulation of a welding process, a postweld heat treatment (PWHT) and a cyclic fatigue loading. The commercial software Virtual Weld Shop is used for the modelling of a welding process and PWHT. Welding induced residual stresses and the information about the material state can then be used in the fatigue analysis of the entire engine structure by the submodeling technique in ANSYS. Based on this calculation procedure a more accurate prediction of the component lifetime is possible. we get the idea of fatigue analysis on multi pass weld from this study [2]

Rabih Kassab, Henri Champliaud* Ngan Van Lê, Marc Thomas in their paper titled "Finite element modelling of a welded T-joint" presented the Finite Element (FE) modelling of a two-seam welding process for a T joint with a V chamfer preparation. The "birth and death" method is used in ANSYS© to simulate the filler metal deposition and the heat source. The weld pool is simulated according to the double ellipsoid configuration proposed by Goldak et al. from the study we get the idea of thermal simulation and the structural simulation of welded joint.[3]

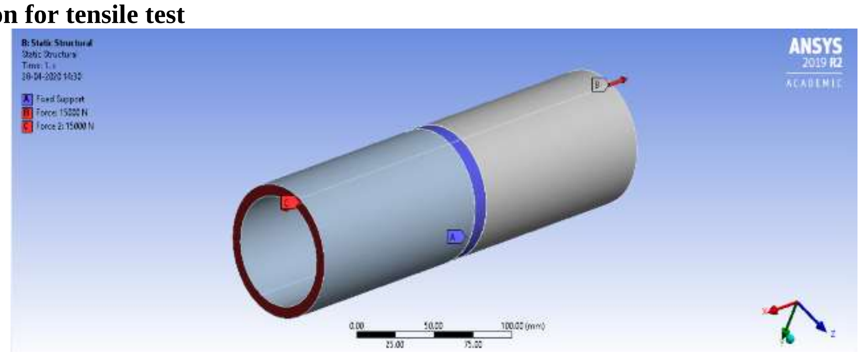
RADU Ioan Eugen1, GREBENIŞAN Gavril1 in their paper titled FINITE ELEMENT ANALYSIS OF A FORMING PROCESS, USING STATIC STRUCTURAL (ANSYS) AND ANSYS LS-DYNA studied that The problem consists on analyse of behaviour of deformed sheet material which got a spatial deformed state, has studied and reported in this work. The analysis method, used here, were Finite Element Analysis, on ANSYS software. We considered that the workpiece was pre-deformed, using a conventional method, by deep forming process (coupled deep drawing- a direct deep drawing combined with an inverse deep spherical forming process). At this stage, the aims were to find most deformed zones, and most exposed (the safety factor magnitude) ones. This work has solved this problems using ANSYS software.[4]

3.Objectives of study
i. Develop CAD Model of welded joint
ii. Define proper loading, boundary and meshing condition for FEA of model
iii. Define FEA process for static structural analysis of weld joint of structural steel and CB08Γ2C

4. Procedure
The model of welded joint is developed using 3d modelling software, solid works was used in this case.

Fig (4.1) CAD model of geometry

After the CAD geometry was developed it was imported in Ansys and two different projects were created. One for tensile testing of joint and one for bent test for joint. CAD model of welded joint was same for both the projects. loading and boundary conditions were different for both tensile and bent test
4.1 Loading and boundary conditions:-
Loading condition for tensile test

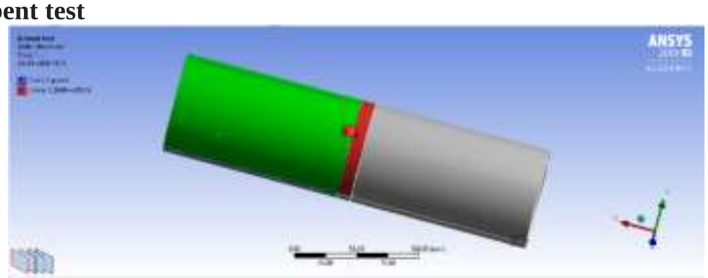


Fig(4.2) tensile test

Table -4.2: tensile test loading and boundary conditions

A	FIXED SUPPORT
FORCE-1	15000 N
FORCE-2	15000 N

Loading condition for bent test



Fig(4.3) Bend test

Table -4.3: bend test loading and boundary conditions

A	FIXED SUPPORT
FORCE-1	1.06*10^5 N

4.2 Material properties
pipe material

pipe material > Constants

Dey	7.786e-005 kg mm^-3

pipe material > Isotropic Elasticity

Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa	Temperature C
2.e+005	0.28	1.5152e+005	78125	

pipe material > Tensile Yield Strength

Tensile Yield Strength MPa
250

pipe material > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
420

weld material

weld material > Constants

Density	7.8e-006 kg mm^-3

weld material > Isotropic Elasticity

Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa	Temperature C
210	0.22	125	86.066	

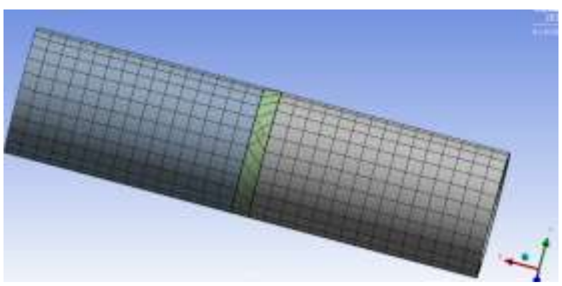
weld material > Tensile Yield Strength

Tensile Yield Strength MPa
450

weld material > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
550

4.3.Meshing

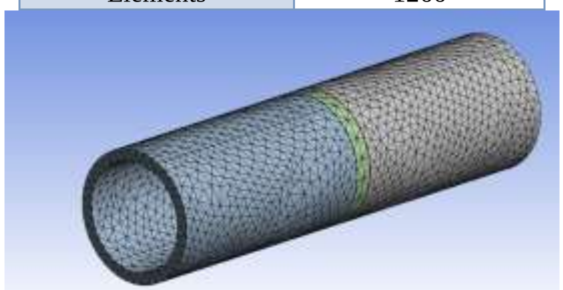


Fig(4.4) meshing for bend test

Table 4.3

Object Name	*Mesh*
State	Solved
Display	
Display Style	Use Geometry Setting
Defaults	

Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	10.0 mm
Sizing	
Use Adaptive Sizing	Yes
Resolution	Default (2)
Mesh Defeaturing	Yes
Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	321.56 mm
Average Surface Area	6769.9 mm²
Minimum Edge Length	3.0 mm
Statistics	
Nodes	6337
Elements	1266



Fig(4.4) meshing for tensile test

Table 4.4 Mesh

Object Name	*Mesh*
State	Solved
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	10.0 mm
Sizing	
Use Adaptive Sizing	Yes
Resolution	Default (2)
Mesh Defeaturing	Yes
Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	321.56 mm
Average Surface Area	6769.9 mm²
Minimum Edge Length	3.0 mm
Quality	
Error Limits	Standard Mechanical
Statistics	

Nodes	6469
Elements	1363

5.RESULTS

Bend test results

TABLE 5.1 Equivalent Stress

Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1.	1.7893	719.94	158.82

FIGURE 5.1 Equivalent Stress > Image

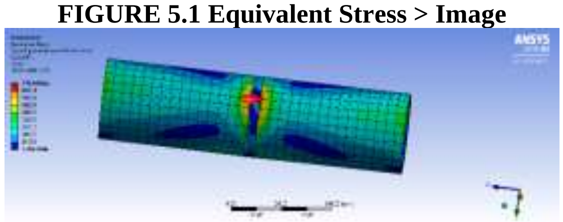


TABLE 5.2 Total Deformation

Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1.	0.	6.1657	0.23557

FIGURE 5.2
Total Deformation > Image

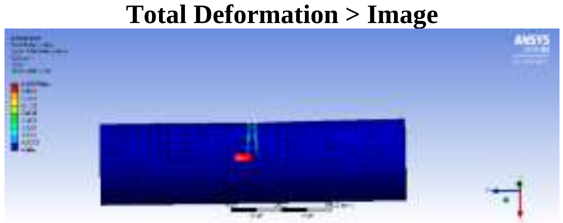


TABLE 5.3
Equivalent Elastic Strain

Time [s]	Minimum [mm/mm]	Maximum [mm/mm]	Average [mm/mm]
1.	2.4214e-004	2.7184	4.6971e-002

FIGURE 5.3
Equivalent Elastic Strain > Image

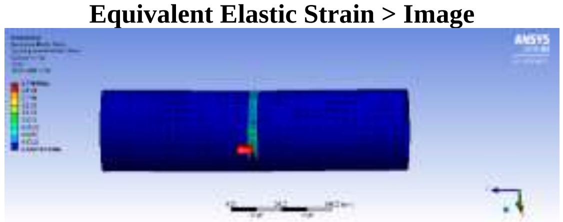
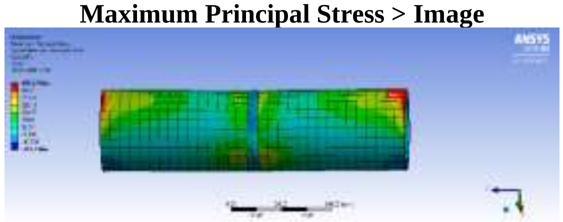


TABLE 5.4
Maximum Principal Stress

Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1.	-107.1	490.27	82.517

FIGURE 5.4
Maximum Principal Stress > Image



Tensile test result

TABLE 5.5 Equivalent Stress

Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1.	209.82	1058.1	499.07

FIGURE 5.5
Equivalent Stress > Image

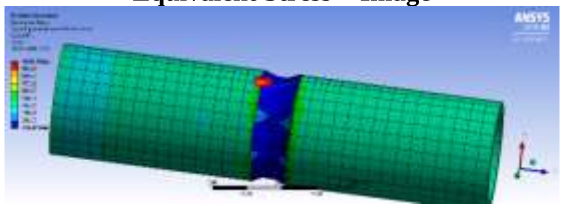


TABLE 5.6
Maximum Principal Stress

Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1.	196.53	1065.5	502.22

FIGURE 5.6

Maximum Principal Stress > Image

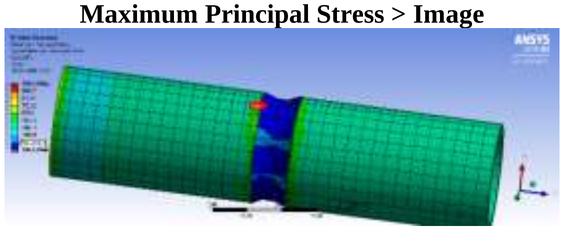


TABLE 5.8
Maximum Principal Stress 2

Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1.	196.53	1065.5	502.22

FIGURE 5.8
Maximum Principal Stress 2 > Image

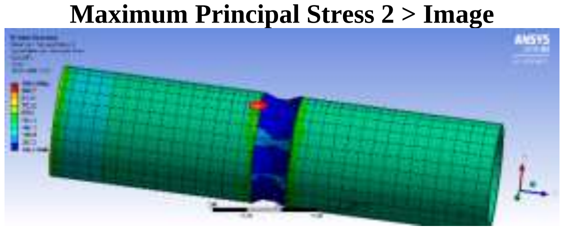


TABLE 5.9
Equivalent Elastic Strain

Time [s]	Minimum [mm/mm]	Maximum [mm/mm]	Average [mm/mm]
1.	1.0805e-003	5.0675	0.19793

FIGURE 5.9
Equivalent Elastic Strain > Image

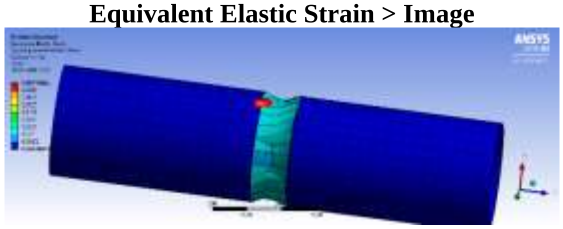
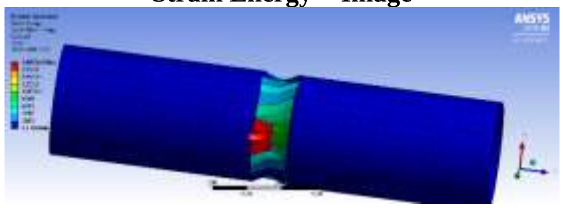


TABLE 5.10 Strain Energy

Time [s]	Minimum [mJ]	Maximum [mJ]	Total [mJ]
1.	5.1318	1.8855e+005	7.0347e+006

FIGURE 5.10
Strain Energy > Image



6.CONCLUSION :

i. Model of welded joint was developed successfully

ii. FEA process for static structural analysis of weld joint of structural steel and CB08Γ2C was developed successfully

6. REFERENCES

[1]Finite element analysis of welded structures .Ramesh Sawant, Sachin Awasare and Asokendu Samanta♣ Long Term Strength Assessment Group Research and Co-ordination Division Indian Register of Shipping, Powai, Mumbai – 400072

[2] On the residual stresses in multi-pass welds: coupling of welding simulation and fatigue analysis A. Krasovskyy*, S. Sönnichsen, D. Bachmann

[3]FINITE ELEMENT MODELING OF A WELDED T-JOINT Rabih Kassab, Henri Champliaud* Ngan Van Lê, Marc Thomas Mechanical Engineering Department (ETS) École de technologie supérieure 1100 Notre-Dame West Montréal, Québec, H3C 1K3 *henri.champliaud@etsmtl.ca Jacques Lanteigne Institut de recherche d'Hydro-Québec (IREQ) 1800, boul. Lionel-Boulet Varennes, Québec, J3X 1S1

[4]FINITE ELEMENT ANALYSIS OF A FORMING PROCESS, USING STATIC STRUCTURAL (ANSYS) AND ANSYS LS-DYNA RADU Ioan Eugen1, GREBENIŞAN Gavril1 1-University of Oradea