GMAW Finite Element Simulation

Luk USA

Kunding Wang, Paul Biggers, Mario Metzger

1.0 Abstract

Gas Metal Arc Welding (GMAW) process is being widely used in the automotive torque converter assembly line for joining and sealing purposes. This paper describes the modeling and simulation of the welding process using the finite element modeling technique. The commercial finite element program ABAQUS/Standard is used to model the 3D thermo-mechanical coupling of the process. The moving heat source, material deposition, temperature-dependent material properties, and large deformations are considered in the modeling. The calculated residual stress is verified by comparing with the stress measurement by X-Ray diffraction. The calculated residual stress is in good agreement with the measurement. Static structural finite element simulation is made with the pressurizing load to calculate the total stress for assessing durability of the baseline weld and a reduced weld for cost saving purpose. The finite element stress analysis shows an insignificant difference value between the residual stress and the total stress. In addition, there is practically no difference in maximum stress between the baseline weld and the reduced weld. The reduced weld is proven to be a feasible cost saving alternative. Six-Sigma process is utilized to develop weld process parameters and preventive measurements are also taken to ensure the desired quality. Potential benefits of future modeling and simulation towards performance and cost are also discussed.

Key Word: Torque Converter, Welding, Residual Stress, Six Sigma

1.0 Introduction

The automotive torque converter that is selected for the study is a fluid-coupling device that also acts as a torque multiplier during initial acceleration. The cover which is the outside half of the torque converter toward the engine side is welded to the pump shell which is the other outside half on the transmission side. The GMAW closure welding is carried out by three robots. Location of the weld is shown in Figure 1.

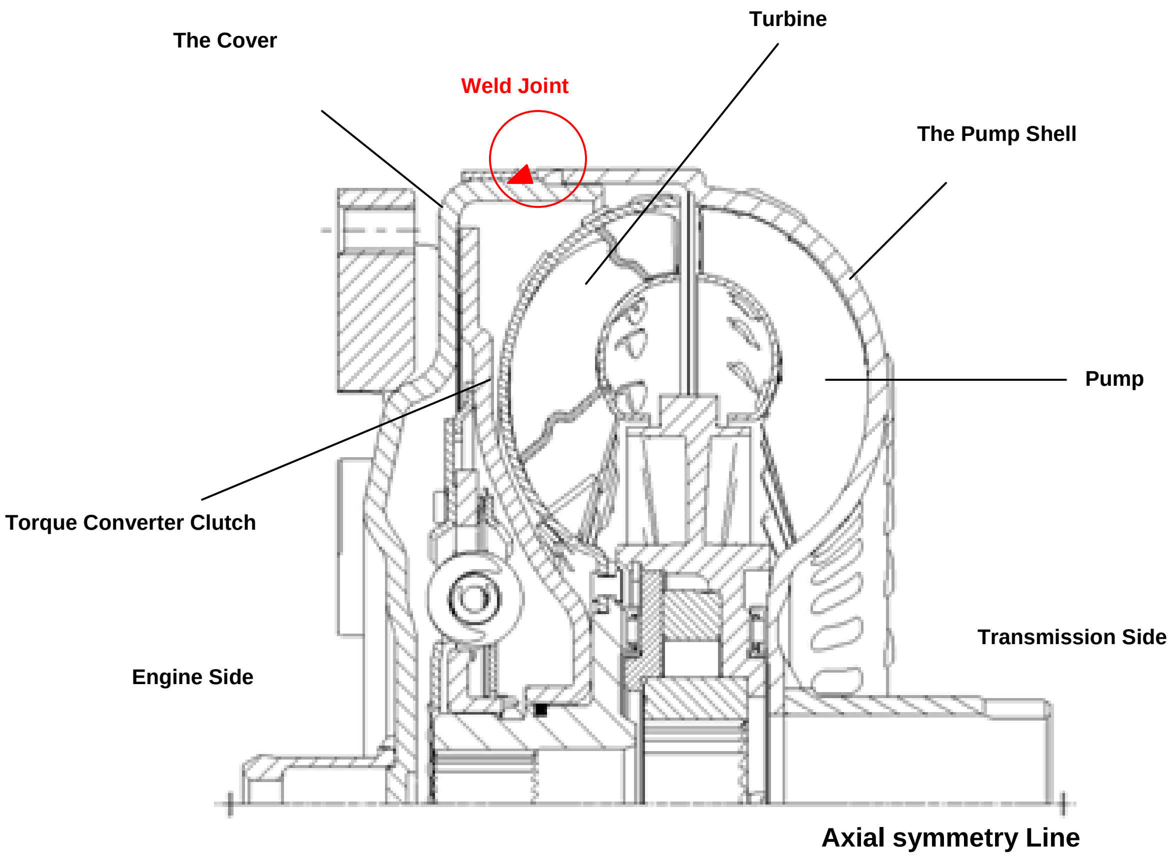


Figure 1. Torque converter

The torque converter is repeatedly under the tremendous pressurized during operation to engage the overdrive in which the torque converter clutch mechanically connects the turbine and the pump to reduce fuel consumption. Figure 2 shows one of the operational modes of the torque converter. The TCC is pushed by the pressurization to engage the cover which is welded to the pump shell. It is extremely important that the weld joint is strong enough to resist the fatigue damage associated with the cyclic pressure.

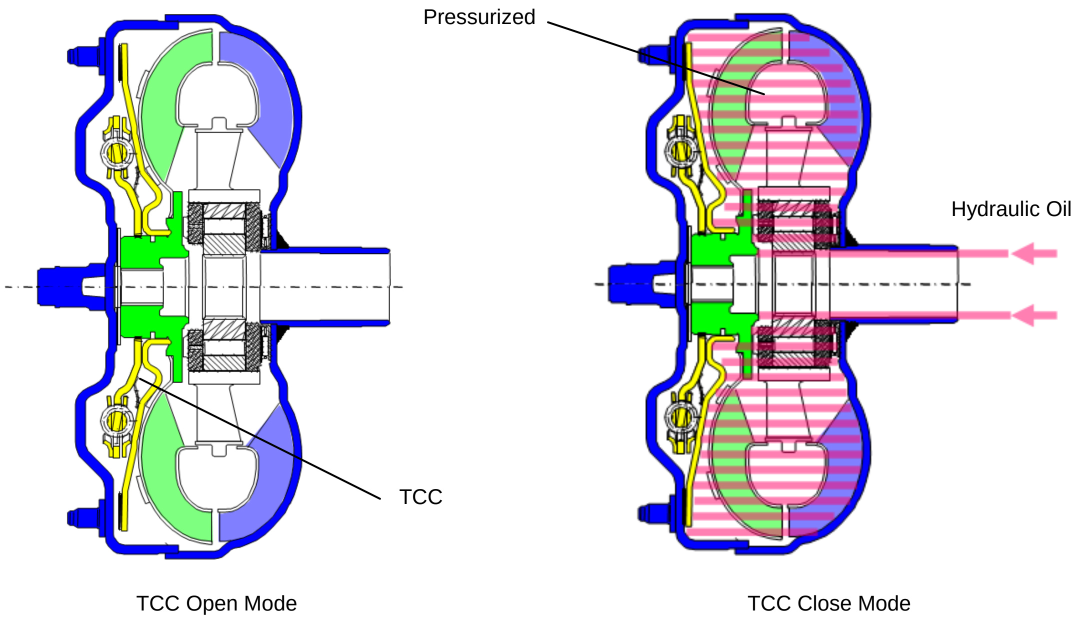


Figure 2. Pressurization of torque converter

One of the important fatigue tests of the torque converter is the pressurizing fatigue test. The objective is to assess the effect of residual stress and working stress on the weld joint durability. The fatigue test is carried out in tensional rotation condition with internal pressure. Figure 2 shows the pressurized area.

2.0 GMAW Welding Process

Most automotive torque converters are assembled together with a GMAW weld. Filler material is heated up by the electric arc to a molten metal state. The molten metal transfers to the weld pool while the gas mixture is provided to shield the weld from nitrogen and oxygen. As the arc moves ahead, the weld pool temperature quickly drops to the room temperature and the molten metal starts to solidify to form a weld line. The heating and cooling cycles cause metallurgical transformation strain (due to the micro-structure changes) and transient thermal strain (due to the volume shrinkage) that are the main causes of residual stress. Residual stress in a welded line is often tensile and, therefore, detrimental with respect to durability. Usually, the residual stress is about on order of the tensile yield strength of the base material. Combination of residual stress and working stress is the main cause of fatigue failure of the weld joint.

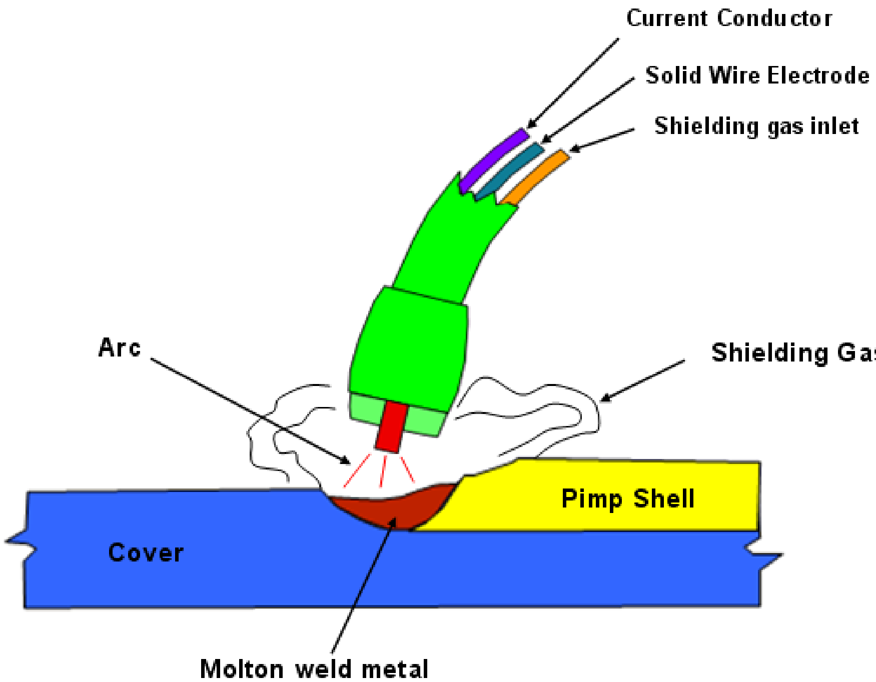


Figure 3. GMAW welding process

There are at least five important GMAW process parameters; arc traveling speed, current, voltage, arc angle, and wire feed speed. In this study, arc traveling speed and current are the major welding process parameters that are controlled to affect geometry and quality of a weld. With a higher arc traveling speed, the baseline weld can be reduced to a smaller size and therefore consumes less filler material. However, if the arc moves too quickly, it can cause insufficient penetration, incomplete fusion, and porosity. It is necessary to balance higher arc traveling speed with a higher current input to ensure weld quality. As can be seen from the normalized parameters in Figure 4, more than 25% weld volume reduction is achieved by combination of a 30% higher arc traveling speed and a higher current input. The reduction of weld volume is translated into a substantial filler material reduction and therefore cost reduction.

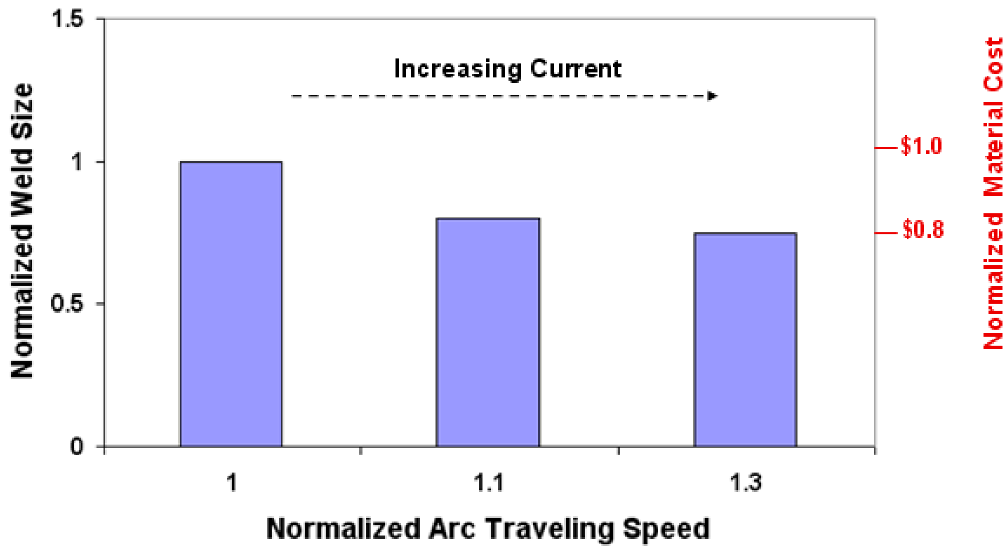
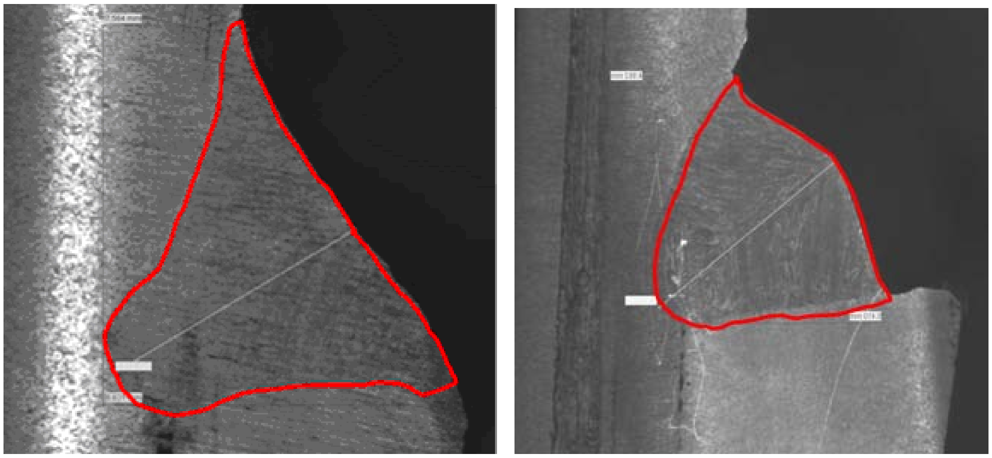


Figure 4. Weld size vs. arc traveling speed

Six-Sigma process is applied to develop the corresponding process parameters and to eliminate metallurgical uncertainties. So, in a defined level of confidence interval, consistent metallurgical properties and dimensional quality are ensured. Prototypes are built for functional evaluation and durability assessment. Preventive measurements are taken to meet the desired specification requirements and metallurgical inspection are made to ensure adequate penetration, complete fusion, and no porosity.

This eliminates the need to develop a costly solidifying simulation in which weld micro-structural transformation change is simulated to calculate its effect on residual stress and strain. The complex thermal, mechanical and metallurgical coupling model is thus reduced to a more efficient thermal-mechanical coupling model. Figure 4 shows respective profile of the baseline weld and the reduced weld. Weld profiles are indicated in red color line. The reduced weld has 30% less volume than the baseline weld does. The baseline and the reduced welds have passed the pressurizing fatigue test requirement.



Baseline Weld Profile Reduced Weld Profile

Figure 5. Weld profiles

4.0 FE Modeling and Simulation

The 3-D Finite element model is built in ABAQUS/Implicit, using 8 node thermal elements for the transient thermal simulation, and 8 node structural elements for the mechanical simulation [1]. The simulation procedures are carried out sequentially; the transient thermal simulation calculates the temperatures, and then is used as input for the mechanical simulation to calculate stress and strain. Changes in the mechanical state are assumed to cause no changes in the thermal state. But changes in the thermal state are assumed to cause changes in the mechanical state. In the mechanical simulation, restart command writes out the residual stress and the displacement to a restart file. Static structural element model imports them as initial conditions, and is applied with operating pressure as loading conditions to calculate total stress for durability assessment. Figure 5 shows the schematic illustration of the simulation steps.

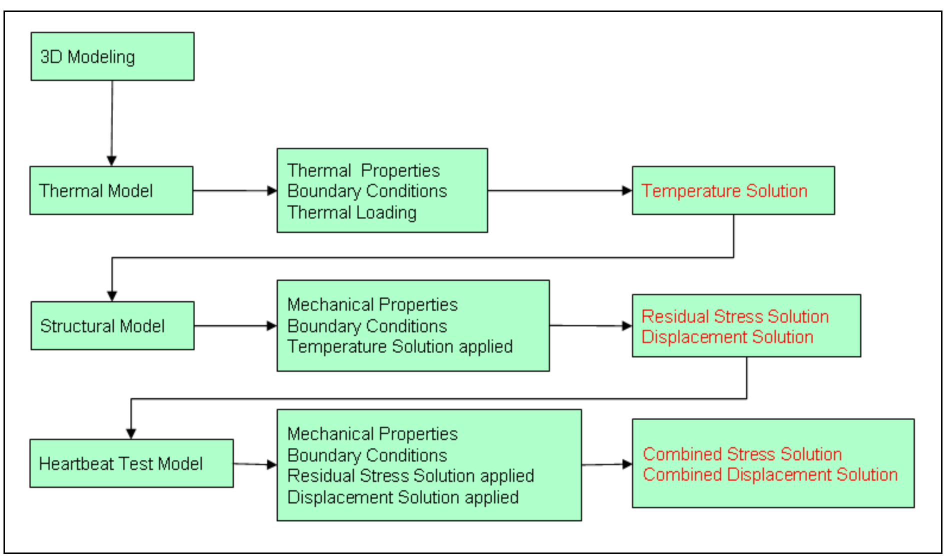


Figure 6. Finite element modeling and simulation

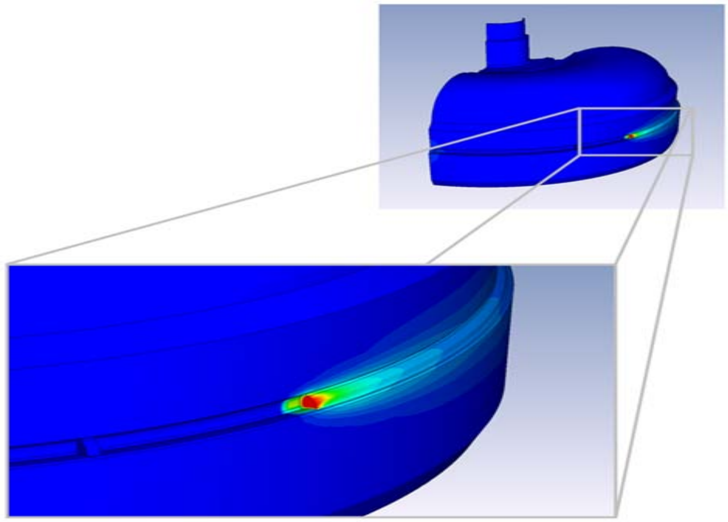


Figure 7. Finite element model

4.1 Boundary Conditions

In the thermal and mechanical simulations, all the nodes at the bottom of the torque converter are clamped. Because there are 3 robots to perform the welding process simultaneously, the modeling takes advantage of the 120^o axis-symmetry to reduce the computational effort. The model is cut down to a 120° model and both end faces are coupled thermally and mechanically to represent the cyclic symmetrical conditions. Figure 7 shows the 120^o cyclic-symmetrical FEA model with a snap shot of transient temperature distribution at the weld line.

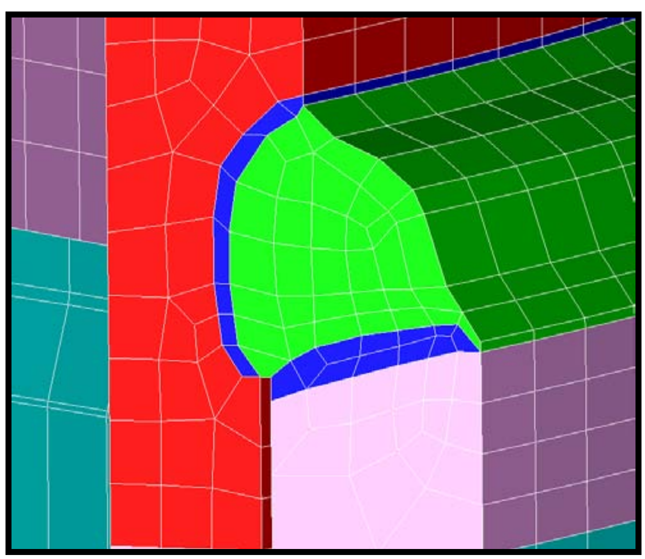


Figure 8. Finite element mesh

Element deactivation and reactivation is utilized to simulate the molten filler material deposition. Body heat flux of the molten filler material is transferred to the base material through the reactivated elements at each time increment. To simulate the traveling arc, the 3-Dimensional moving heat source [2] is programmed in the user subroutine to simulate surface heat flux at the same time increment. Shield gas is modeled as surface heat flux [3]. Distribution and magnitude of the heat flux are also defined and programmed in a user subroutine. To simulate the heat energy loss to the air through the torque converter’s free surfaces, its outer and inner surfaces are assigned as convective surfaces. The film coefficient is assumed to be constant throughout the surfaces. Static structural finite element model is developed to simulate the effect of pressurization of torque converter on weld stress for durability assessment. The model is applied with the operating pressure as loading condition. End faces are mechanically coupled to represent a 120^o cyclic symmetrical condition.

4.2 Material Model

For the thermal modeling, thermal conductivity, specific heat, material density, latent heat, and Stefan Boltzmann constants [4] are provided. These values are programmed to be temperature-dependent in the form of a table. Temperature-dependent mechanical properties are also provided for the mechanical modeling. Three different material properties are involved in the calculation. The pump shell consists of common low alloy steel, the cover is made of high-strength low-alloy (HSLA) steel, and the weld is made out of low carbon casting material. Stress-strain curves for these materials are programmed [5] to be dependent upon temperature. Thermal expansion is also defined to be dependent upon temperature change in a user subroutine. In Figure 8, the weld seam is defined by the green colored elements and the heat affecting zone (HAZ) is defined by the blue colored elements. Low carbon casting material is defined for both the weld seam and the HAZ.

5.0 Simulation Results

The reduced weld, which has a smaller profile than the baseline weld geometry is considered to be a cost reduction alternative. Small weld geometry is achieved by increasing arc traveling speed while current is so adjusted to control quality. Two separate FE models are made to simulate both welds respectively for residual stress. Closely matched mesh pattern and density is used for both models for a fair comparison. Simulation starts with thermal simulation for the temperature and then used as initial condition for mechanical simulation to calculate residual stress and strain. In order to verify the mechanical model, the calculated residual stresses at selected locations are compared with stresses measurements by X-Ray diffraction. Good agreement is concluded for the comparison. The comparison is shown in Figure 9 for reference. A relatively sharp gradient is found at the locations D and E. Residual stress falls off sharply from the high tensile stress to the low compressive stress. Thermal contraction that causes highly localized volume shrinkage at the weld is the main cause of residual stress.

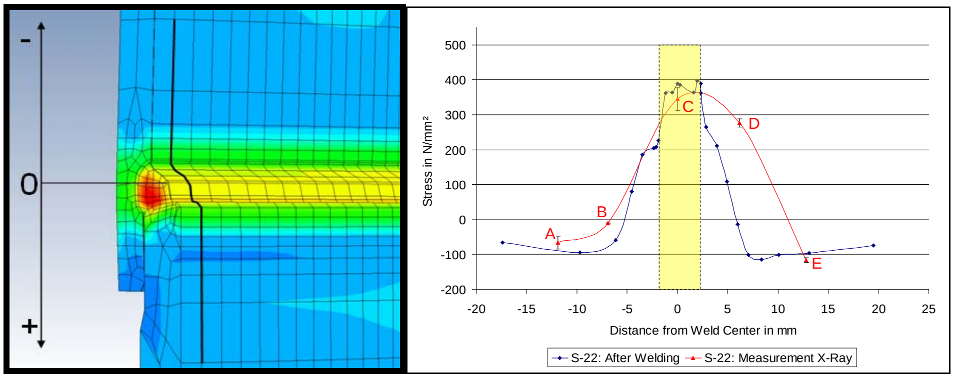
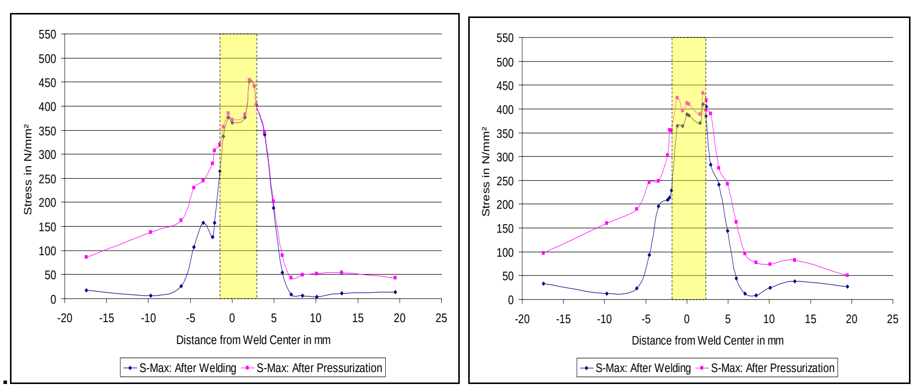


Figure 9 Comparison FEA and X-Ray diffraction

The calculated residual stresses show a good trend of axis-symmetry along the circumferential welding direction for the hoop and the radial residual stresses, except at the welding start and stop position. The calculated residual stress is about in the order of the tensile yield strength of the base material. The predicted residual stress is then used as initial stress in the static structural finite element simulation for the durability assessment. Operating pressure is applied as loading conditions to calculate total stress. Results of the simulation show that very small amount of difference in stress is found between the total stress and the residual stress (Figure 9). Mainly because the combined effect of both applied stress and high initial residual stress causes shakedown of the residual stresses. Therefore durability of both welds should be practically the same. Finally, the preload that is associated with residual stress can limit radial displacement and axial expansion of the torque converter when it is applied with the operating pressure.



Baseline Weld - Stress Results Reduced Weld - Stress Results

Figure 10 Stress results comparison

5.0 Conclusion

3-D finite element modeling and simulation is successfully developed, and is applied effectively to calculate residual stress of an automotive torque converter closure weld by GMAW welding process. Residual stresses are axis-symmetry along the circumferential welding direction except at the start and stop location. Residual stress is about equal to tensile yield strength of the base material. Very small difference in stress is found between total stress and residual stress because of shakedown of residual stress. Durability of both welds is practically the same. Thus, the reduced weld is considered a cost saving alternative. Six-Sigma process eliminates the need of a costly solidifying simulation. The GMAW modeling can be reduced to a more efficient transient thermal-mechanical coupled modeling.

It is recommended to further eliminate the need to use the thermal simulation to calculate the transient temperature what can also be determined by the experimental data. In so doing, the GMAW welding simulation times can be reduced significantly and the use of static structural finite element simulation permits non-FEA engineer to perform weld stress analysis. Six-Sigma process is applied to develop the corresponding welding process parameters, S-N curve and take into account the material hardness. Finally, effects of the residual stress relaxation with 3-D welding finite element simulation should be investigated further to help improving weld design/process optimization.

6.0 References

[1] Finite Element Analysis of Heat Flow in Single-Pass Arc Welds
E.A. Bonifaz

[2] A New Finite Element Model for Welding Heat Source. Metallurgical Transactions
J. A. Godak and A. P. Chakravarti

[3] Introduction to Heat Transfer, Third Edition
Frank P. Incorporeal and David P. DeWitt

[4] ASM Handbook Volume 6 Welding, Brazing and Soldering.
ISBN:087170-382-3

[5] Abaqus Online Documentation. Version 6 5-1