**ME8373 Spring 2015 ICME Proposal**

ICME Analysis of Fatigue Crack Growth through a

Weld in SA-516 Grade 70 Plate

Jody L. Woods

NASA, John C. Stennis Space Center
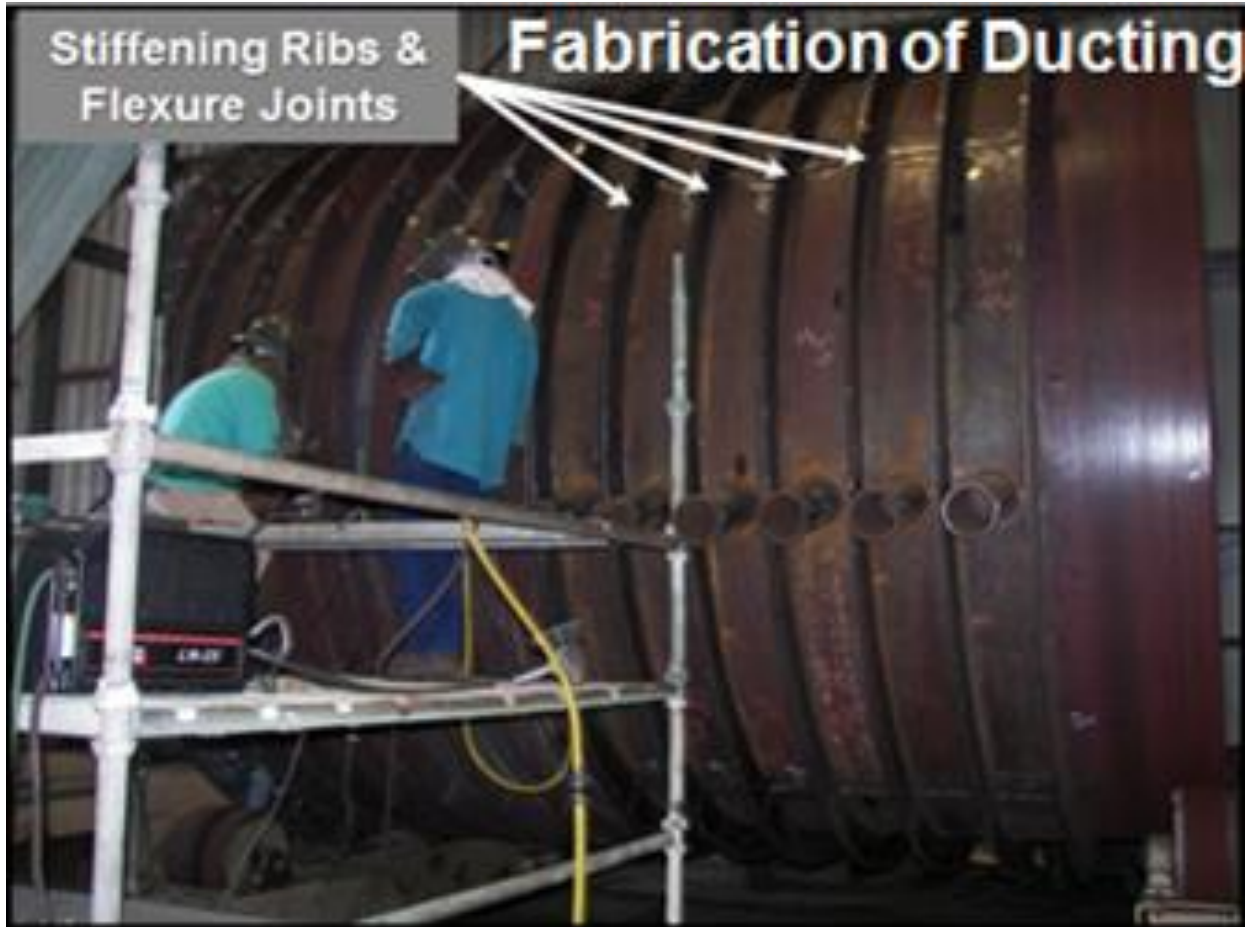
Stennis Space Center, MS

February 26, 2015

Introduction

This paper describes work accomplished to predict the service life of a flexure joint design which is a component of a diffuser duct in the A3 Test Stand, an altitude simulation rocket engine test facility at NASA's Stennis Space Center. The duct has two pressure shells separated by cooling water passages and connected by stiffening ribs and flexure joints. Rocket exhaust flows within the duct and heats the inner pressure shell while the outer pressure shell remains at ambient temperature. The flexure joints allow for differential thermal expansion of the inner and outer pressure shells and are subject to in-service loading by this thermal expansion along with water pressure in the cooling water passage, atmospheric pressure outside the duct, near vacuum conditions within the duct, and vibrational loads from operation of the facility and rocket engine. Figure 1 shows a schematic axisymmetric cross section of the diffuser pressure shells and flexure joints with a zoomed in view of the flexure joint. The flexure joints are expected to eventually fail by fatigue cracking leading to leaks from the cooling water passages to the outside. The zoomed in view in Figure 1 indicates where cracking is expected to occur, namely through a weld bead between two plates of SA-516 Grade 70 steel. This weld bead acts as the fulcrum of the flexure joint and it is clear from inspection of the geometry and loading represented in the zoomed in portion of Figure 1 that inherent in the design there is a severe notch formed between the flexure plate, weld bead, and stiffening ring that will be the site of crack initiation and location from which the crack grows to the outer surface of the weld bead.
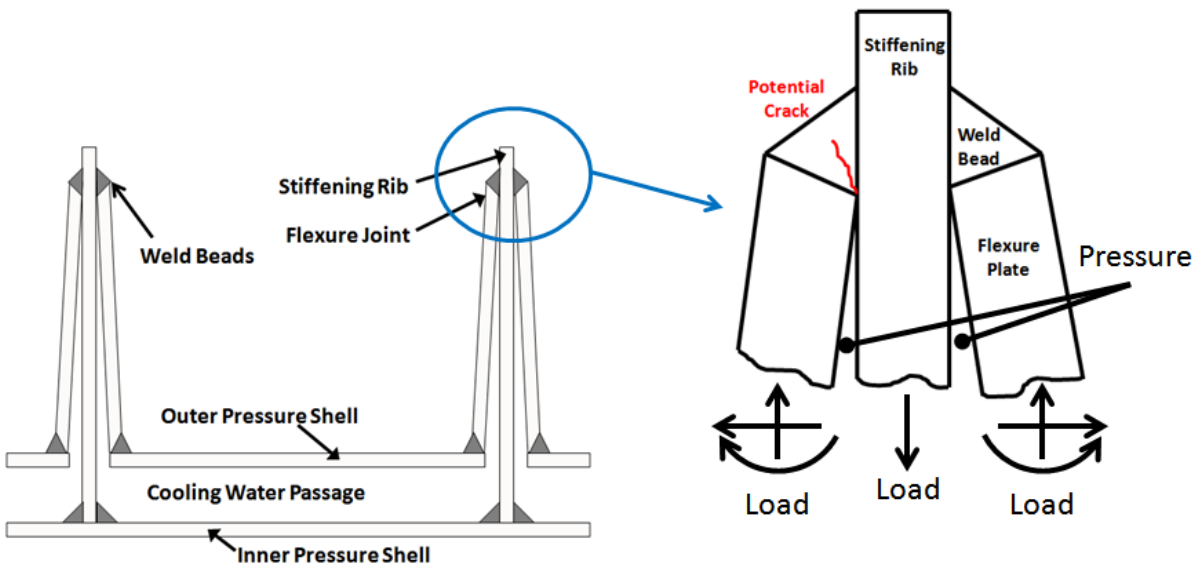
Figure 1: Flexure Joint Potential Cracking

Figure 2 is a photograph of a section of the diffuser duct taken during its fabrication showing locations of stiffening ribs and flexure joints. The entire length of duct in the diffuser system is approximately three-hundred-fifty feet and depending on location along the system, the duct is between eleven and seventeen feet in diameter. There are two flexure joint designs present in the diffuser system, one is a low heat flux design with one foot spacing between stiffening ribs and the other is a high heat flux design with three inches between ribs. Although the stiffening rib spacing as well as their other dimensions differ, the two designs are topologically identical. The high heat flux design is located near the diffuser entrance where rocket exhaust impinges directly on ducting. The rest of the diffuser including the duct section shown in Figure 2 is the low heat flux design. As there is one flexure joint located on either side of each stiffening rib, the entire diffuser system contains over a thousand of these flexure joints. This equates to over seven miles of flexure joint weld bead to be concerned with fatiguing and developing leaks.

Figure 2: Diffuser Duct Fabrication Photograph

As previously mentioned, this diffuser duct is part of the A3 Test Stand, a NASA altitude simulation rocket engine test facility at Stennis Space Center. A photo of the A3 Test Stand with the diffuser installed is shown in Figure 3. Only a portion of the diffuser is visible laying horizontal to the left of the test stand. There is also an elbow and a vertical portion inside the test stand structure and obscured in the photo. Note the truck pointed out in the photo for a sense of the scale of the facility.
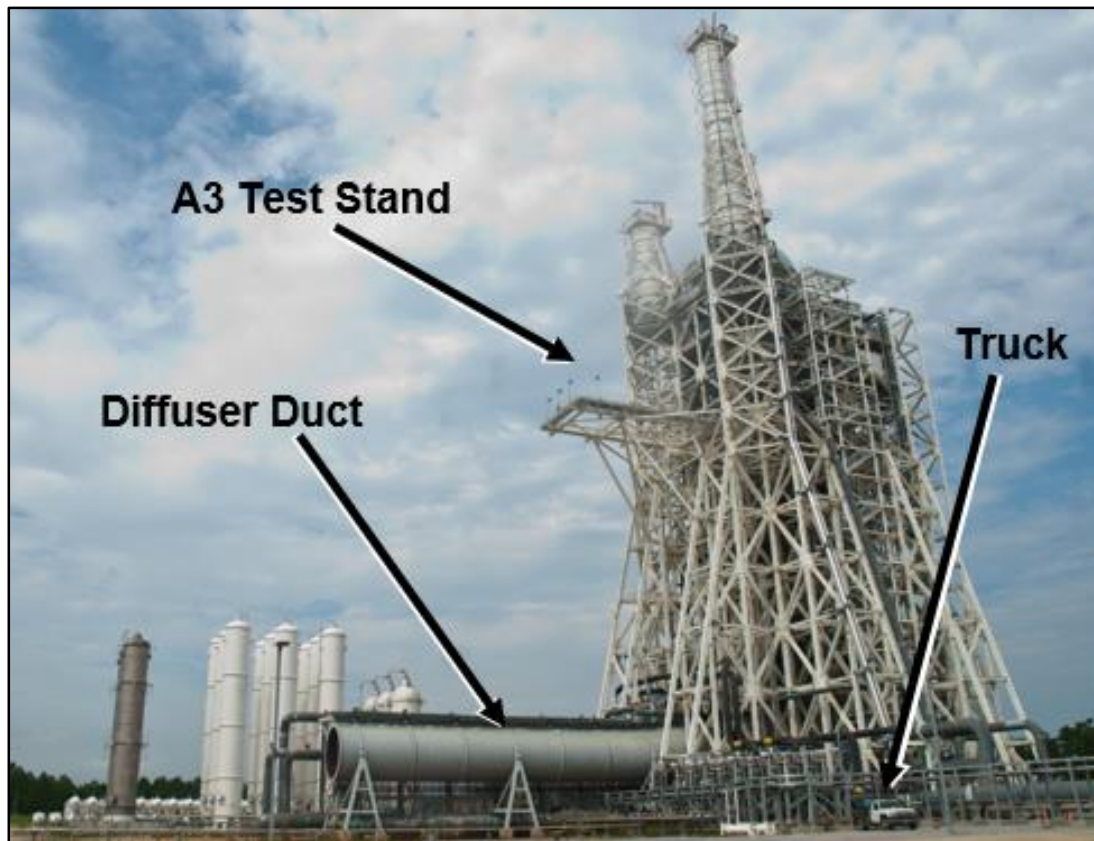
Figure 3: Diffuser Duct Installed in A3 Test Stand

Construction of the A3 Test Stand has just recently completed and it has not been placed in service yet. It is almost a certainty that this flexure joint will eventually fatigue and leaks will develop from the cooling water passages. When this happens, maintenance will be required to weld-repair the leaks. It is important to have an estimate of when this will begin to occur as unanticipated downtime for maintenance during an active rocket engine test program is problematic. If it could be anticipated when repairs would begin to be required, time and money for that can be built into project schedules. On the other hand, building in schedule contingency if it is not really required would be inefficient project management which could be avoided as well with an accurate estimate of when maintenance to repair leaks will begin to be required.

Requirements for Development of the Service Life Prediction Model

The certainty that the fatigue being investigated will occur in a weld in the relatively complex combination of geometry and loading of the flexure joint makes prediction of service life rather complex. This is not a laboratory specimen yet published fatigue data is typically for laboratory fatigue test specimens. The model developed to make the life prediction must take into account the effect of fatigue occurring in a weld in terms of variation of the microstructure within the weld due to differences in solidification and cooling rates at different locations within the weld during the welding process and potential presence of defects and inclusions. It has been shown that the variation in microstructure in a weld and weld affected zone in low carbon steels has a definite effect on tensile strength, fracture toughness, ductility, and fatigue strength [1-2]. It has also been shown that larger grain size in the weld and weld affected zone results in reduced fatigue crack growth rate for a given stress intensity range [3]. Even though the weld affected zone microstructure has been shown to influence fatigue properties, it should be noted that it has also been shown that the effect is minor when compared to the effects of weld geometry [4]. Thus the model developed to make the life prediction must also must take into account the flexure joint's geometry, as well as in-service loading, which together define the force driving crack propagation. Finally there must be a way to validate the prediction of life under in-service loading and quantify the uncertainty in the prediction through assessment of uncertainty in the model's inputs and its effect on the results as well as some comparison of the model's results to relevant experimental data.

Integrated Computational Materials Engineering Approach

Development of the model used to predict the service life of the flexure joint adopted an Integrated Computational Materials Engineering (ICME) approach. This approach employs a hierarchical multiscale modeling methodology using simulations at various length scales that are run in an integrated fashion with data bridges between the simulations and the whole suite then solves the overall engineering problem at hand [5]. The data bridges involve downscaling of data requirements and in some cases, input data from higher length scale simulations to lower length scale ones and then upscaling of output data from the lower scale simulations to serve as input for the higher scale ones. The implementation of this approach for the problem of predicting the flexure joint service life is illustrated in Figure 4. The individual simulations and data bridges along with the associated length scales are indicated in the figure and described in the following paragraphs. The data bridge numbering follows the convention set forth in [5].
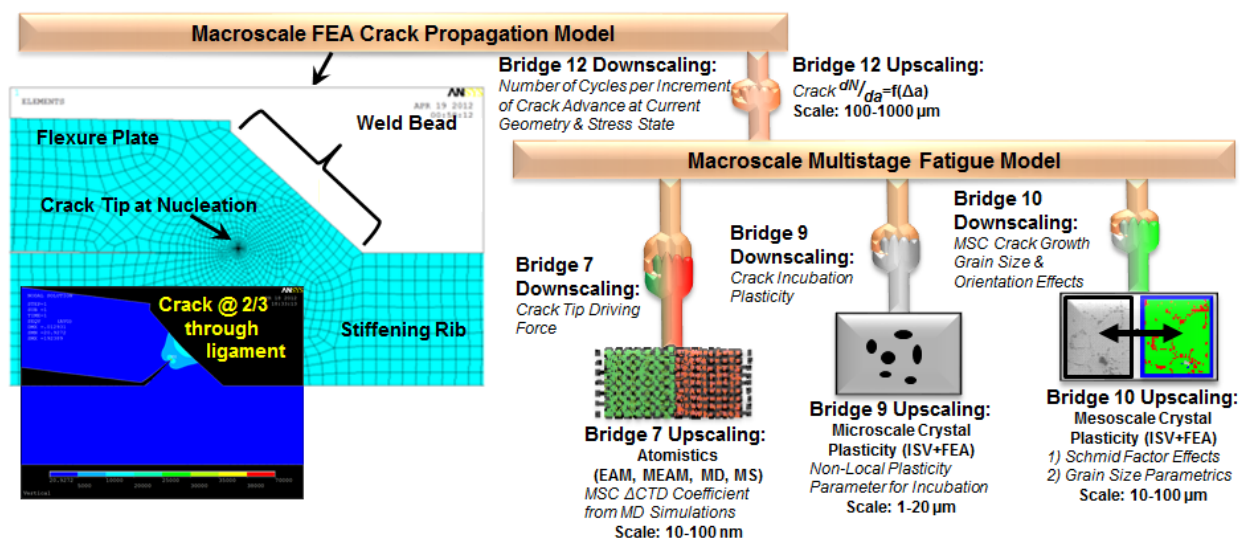

Figure 4: ICME Modeling Approach for Flexure Joint Fatigue Simulation

At the highest abstraction level and encompassing the entire scale of the flexure joint, a finite element model was developed specifically for this problem that incorporates the flexure joint geometry, bulk material properties, loads, and boundary conditions. This model is denoted as "Macroscale FEA Crack Propagation Model" in Figure 4 and it utilizes an algorithm developed based on Fracture Mechanics and Finite Element Analysis techniques [6-7] to incrementally grow a crack starting at the severe notch between the flexure plate, weld bead, and stiffening rib and growing along the path dictated by the evolution of the state of stress local to the crack tip as the crack grows and changes the domain geometry and thus the load path through the weld bead. The algorithm grows the crack incrementally at a set distance per increment, on the order of a tenth to one millimeter. In order to make a prediction of service life, the simulation needs the number of loading cycles associated with each increment of crack growth. This data requirement along with the state of stress local to the crack tip is the downscaling information for bridge twelve which connects the Macroscale FEA Crack Propagation Model to the Macroscale Multistage Fatigue Model at the length scale of a tenth to one millimeter. The Macroscale Multistage Fatigue Model calculates the number of loading cycles for each increment of crack growth and this is the upscaling information for bridge twelve. Once the Macroscale FEA Crack Propagation Model has grown the crack all the way through the weld bead, the sum total of loading cycles calculated in this manner defines the service life.

The Macroscale Multistage Fatigue Model was developed based on the work of Horstemeyer et al [8-11] and simulates fatigue based on different stages in the evolution of a fatigue crack. The stages are crack incubation, microstructurally small crack,

physically small crack, and large crack. Each of the stages has a different mathematical framework upon which the simulation is based. As the Macroscale FEA Crack Propagation Model relies on information from the Macroscale Multistage Fatigue Model, the Multistage Fatigue Model in turn relies on information provided by lower length scale based simulations. The inputs required by the Multistage Fatigue Model include microstructurally small crack growth grain size and orientation effects which is the downscaling information for bridge ten in Figure 4. Information on crack incubation plasticity is also required and is the downscaling information for bridge nine. Finally, information on the crack tip driving force is required and is the downscaling information for bridge seven.

The microstructurally small crack growth grain size and orientation effects information required for bridge ten is calculated using a Mesoscale Crystal Plasticity Model at the length scale of ten to one hundred micrometers that combines Finite Element Analysis with Internal State Variables and upscales its data requirement to the Multistage Fatigue Model as Schmid Factor Effects and Grain Size Parametrics. Inputs to the Mesoscale Crystal Plasticity Model include information about the actual microstructure being simulated. Samples of the Flexure Joints were procured for testing as will be discussed in more detail in the next section. Some of these samples are used for and microstructural and metallurgical evaluation to provide input to the individual ICME simulations such as the microstructural information required as input to the Mesoscale Crystal Plasticity Model being discussed.

The crack incubation plasticity information required for bridge nine is calculated using a Microscale Crystal Plasticity Model at the length scale of one to twenty micrometers that combines Finite Element Analysis with Internal State Variables and upscales its data requirement to the Multistage Fatigue Model as a Nonlocal Plasticity Parameter for Crack Incubation. Inputs to the Microscale Crystal Plasticity Model include information about the actual microstructure being simulated and, as with the Mesoscale Crystal Plasticity Model, these are provided by evaluation of the test samples.

The crack tip driving force information required for bridge seven is calculated using a Molecular Dynamics Simulation at the length scale of ten to one hundred nanometers. This simulation upscales its data requirement to the Multistage Fatigue Model as a Microstructurally Small Crack Delta Crack Tip Displacement Coefficient. Inputs to the Microscale Crystal Plasticity Model include information about the material composition being simulated and these are also provided by evaluation of the test samples.

Quantification of Uncertainty and Experimental Validation of Model Results

In order to have confidence in the accuracy of and, more importantly, the limitations on the precision of the ICME model's prediction of flexure joint service life, efforts were undertaken to quantify the uncertainty in the prediction based on the uncertainty in the inputs to the model [12]. Furthermore, the ICME model's predictive capability and estimate of uncertainty in its predictions was validated through comparison of model results to relevant experimental data.

Specimens intended for fatigue testing that represent the two flexure joint designs were purchased during the A3 Test Stand Construction Project. The geometry of the low heat flux design specimens is shown in Figure 5 and that of the high heat flux design specimens is shown in Figure 6. One-hundred-twenty specimens of each design were purchased. The specimens were fabricated by the same contractor that built the diffuser ducting. They were fabricated using the same materials, tooling, procedures, and by the same workers as were used in fabrication of the diffuser ducting. They were made just like the diffuser ducting was made except that once a section was complete it was sliced into segments as specified in Figures 5 and 6. Figure 7 is a photograph of the test specimen shipping crate opened and with two of the low heat flux and four of the high heat flux specimens placed on top of the crate.
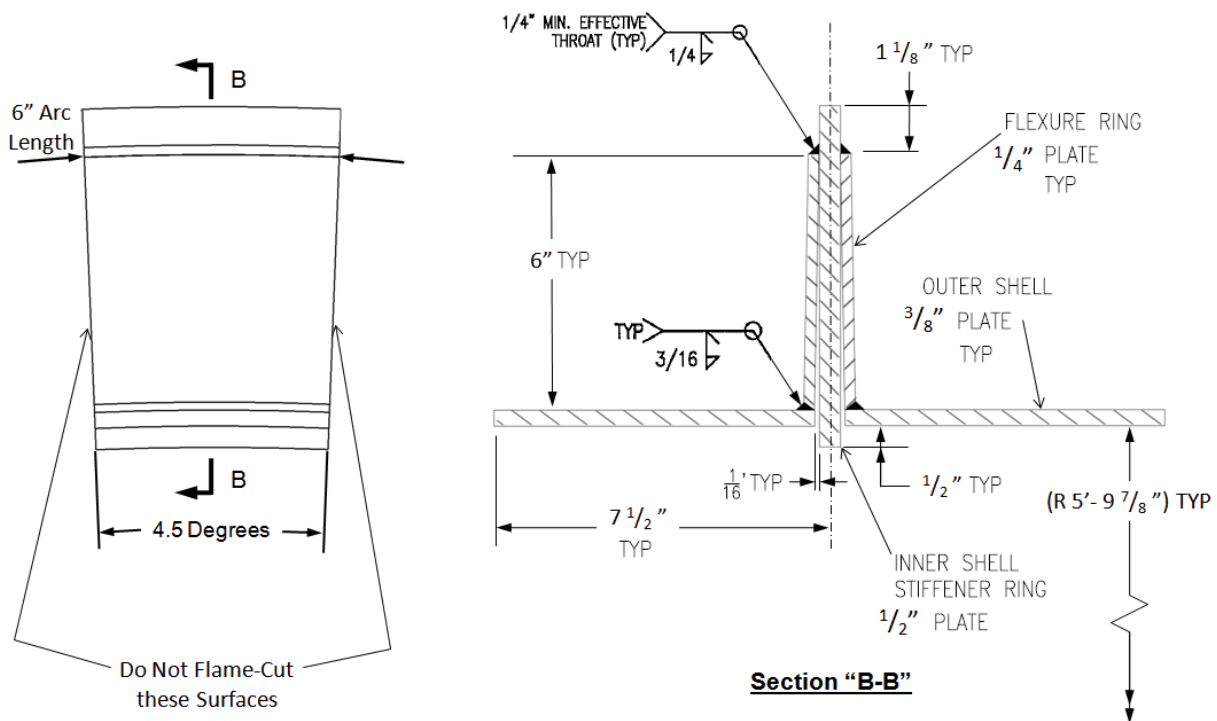
Figure 5: Low Heat Flux Design Flexure Joint Fatigue Test Specimen Geometry
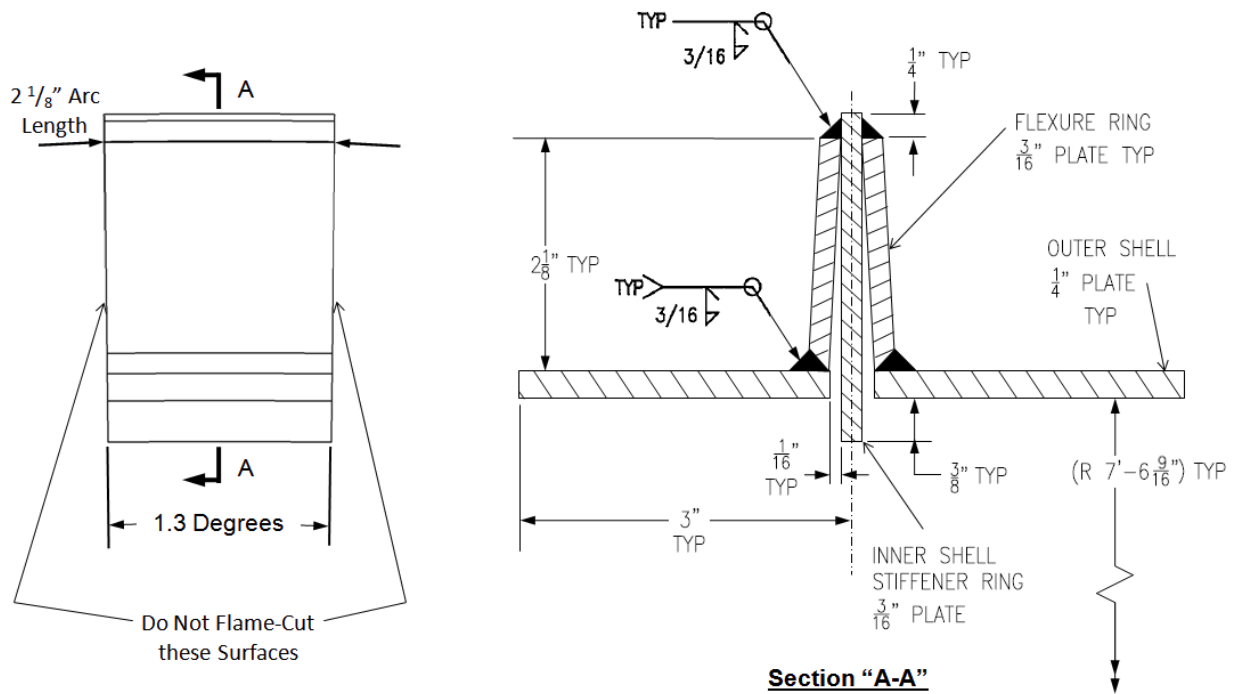
Figure 6: High Heat Flux Design Flexure Joint Fatigue Test Specimen Geometry

Figure 7: Flexure Joint Fatigue Test Specimens

Out of the entire set of test specimens, six of each design were reserved for metallurgical and microstructural analysis to determine the required inputs for the ICME model. In order to facilitate the uncertainty analysis of the ICME model predictions, each input for the ICME model determined through the metallurgical and microstructural analysis is not treated as a discrete value. Rather it is a derived range of values that takes into account the variability of quantities of interest observed in the specimens as well as the uncertainty in the derived quantities. This allows for a matrix of ICME model inputs to be developed. Although for a single life prediction the inputs are discrete values, with this approach a predictions are made that cover the entire input space and the resulting range of ICME model life predictions allow for an estimate of prediction uncertainty and variability, or scatter, in service life due to the physical variability in the seven miles of flexure joint weld bead in question.

The remaining test specimens were used for fatigue testing to validate the ICME model's predictive capability. Fatigue testing by itself using these specimens could not have been used to predict the flexure joint service life. The in-service loading imposed on the flexure joint is complex and would be very hard to duplicate in a laboratory setting, yet can straightforwardly be accomplished virtually in the ICME model. So in order to validate the model's predictive capability for in-service loading, it was used to make predictions for fatigue life of the test specimens subject to a set laboratory loading conditions that bounds the expected in-service loading. The validation of the predictive capability then comes from comparison of the predicted fatigue lives, and predicted scatter in life, for the set of laboratory loading conditions to the observed fatigue lives of the test specimens. The

laboratory loading consisted of tests run under displacement control and tests run under load control. The in-service loading is actually a superposition of displacement and load control components. Additionally for each loading condition tests were run at a "low", "medium" and "high" load or displacement level. This results in nineteen specimens tested at each discreet combination of load control and load level. As already mentioned, the low, medium, and high load and displacement levels were chosen to bound the expected in-service loading.

References

1) Pang, W., Ahmed, N., & Dunne, D., 2011, "Hardness and microstructural gradients in the heat affected zone of welded low-carbon quenched and tempered steels", Australasian Welding Journal, 56, pp. 36-48.

2) McGrath, J., Chandel, R., Orr, R., & Gianetto, J., 1988, "Microstructural / Mechanical Property Relationships in Thick-Section C-Mn Narrow Groove Welds", American Welding Society Welding Journal – Welding Research Supplement, pp196s-201s.

3) Kusko, C.S., DuPont, J.N., & Marder, A.R., 2004, "The Influence of Microstructure on Fatigue Crack Propagation Behavior of Stainless Steel Welds", American Welding Society Welding Journal, 83, pp. 6-s to 14-s.

4) Pollard, B., & Cover, R., 1972, "Fatigue of Steel Weldments", American Welding Society Welding Journal – Welding Research Supplement 51, pp. 544-s to 554-s.

5) Horstemeyer, M. F., 2012, "Integrated Computational Materials Engineering (ICME) for Metals", John Wiley & Sons, New Jersey, ISBN: 978-1-118-02252-8, pp. 1-30.

6) T.L. Anderson, 1994, "Fracture Mechanics Fundamentals and Applications", 2nd Ed., CRC Press, Taylor & Francis Group, Boca Raton, FL, ISBN 0849342600

7) Strain, J., Miller, E., 2013, "Introduction to the ANSYS Parametric Design Language (APDL): A Guide to the ANSYS Parametric Design Language", CreateSpace Independent Publishing Platform, USA, ISBN 146641135X 9781466411357.

8)   McDowell DL, Gall K, Horstemeyer MF, and Fan J, 2003, "Microstructure-Based Fatigue Modeling of Cast A356-T6 Alloy," Engineering Fracture Mechanics, Vol. 70, pp.49-80.

9)   Xue Y, El Kadiri H, Horstemeyer MF, and Weiland H, 2007, "Micromechanisms of multistage fatigue crack growth in a high-strength aluminum alloy," Acta Materialia Elsevier Ltd, Oxford, OX5 1GB, United Kingdom, 55(6) pp.1975-1984.

10) Xue Y, McDowell DL, Horstemeyer MF, Dale MH, and Jordon B, 2007, "Microstructure-based multistage fatigue modeling of a 7075-T651 aluminum alloy," Engineering Fracture Mechanics, 74, pp. 2810-2823.

11) Johnston S, Potirniche GP, Daniewicz SR, and Horstemeyer MF, 2006, "Three-Dimensional Finite Element Simulations of Microstructurally Small Fatigue Crack Growth in 7075 aluminum alloy," Fatigue Fracture Engineering Materials Structures, Vol. 29, pp. 597-605.

12) Coleman, HW, Steele, G, 2009, "Experimentation, Validation, and Uncertainty Analysis for Engineers, 3rd Edition", John Wiley & Sons, New Jersey, ISBN: 978-0-470-16888-2, pp. 193-215.