15th International Conference on Fracture (ICF15)
June 11-16, 2023 Atlanta, GA USA

3-D CRACK MODELING CASE STUDIES FOR FITNESS-FOR-SERVICE ASSESSMENT USING WARP3D AND FEACRACK

Greg Thorwald[1]* and Ted Anderson[2]

[1]Quest Integrity, Boulder, CO, USA, [2]TLAnderson Consulting, Cape Coral, FL, USA
** Presenting Author email: G.Thorwald@QuestIntegrity.com*

Abstract

Two case studies are presented to investigate the use of 3-D crack meshes in Warp3D FEA for FFS assessments. A focused type crack mesh is used for ductile tearing analysis to determine if a cylinder model can be used as approximation for a pipe elbow. The limiting flaw curve results comparison shows an elbow model is needed to obtain more accurate results. A cell type crack mesh is used to examine pipeline hydrotest pressures to cause fatigue crack retardation by crack face closure. The test pressure giving the most benefit is determined by comparing crack closure results from several load cases.

1. Introduction

Fracture mechanics analyses are useful in Fitness-for-Service (FFS) assessments of damaged equipment to determine if the equipment is acceptable to remain in service in its current condition, needs to be monitored, needs to be repaired, or needs to be replaced. The API 579-1/ASME FFS-1 standard Part 9 provides crack analysis guidelines. Finite Element Analysis (FEA), which employs 3-D crack meshes, computes the fracture mechanics values needed for the FFS assessment of particular equipment geometry with a crack. The type of crack mesh used depends on the analysis. Two case studies are discussed to examine the model and analysis details.

In these case studies, a combination of the Warp3D FEA solver and FEACrack mesh generator are used to compute the crack front values. The FEACrack software generates the 3-D crack mesh and writes the Warp3D input file. The Warp3D FEA solver computes the usual displacement, stress, and strain results, along with the crack front J-integral values for the specified number of contours around the crack front. FEACrack is used for post-processing to read the Warp3D results file, display the mesh with results, and plot the J-integral crack front values. Quick post-processing of large results files is supported by the Warp3D binary packets file format, which has a dense format that can be read quickly.

2. Results

For the first example, does a straight cylinder give a good approximation of a piping elbow with an axial crack to allow a rapid fracture mechanis FFS assessment? Critical flaw sizes were determined using 12 crack models: three crack length to crack depth aspect ratios (c/a) and four crack depth to thickness ratios (a/t). Figure 1 shows the focused type crack mesh used for the elastic-plastic FEA to compute crack front J-integral values. The results give a smaller limiting crack size trend for the elbow versus the straight pipe, showing the straight pipe is not a good approximation for the elbow in this case. The overall modeling and post-processing effort took about 16 hours, including creating the custom elbow geometry, creating the crack meshes and input files, and extracting results for post-processing and the ductile tearing analysis. The FEA run-time is not included as the models were run overnight. The modeling time shows it is possible to use advanced analysis methods for practical FFS assessments.

For the second example, hydrostatic testing of pipelines can alter the characteristics of flaws that survive the test, which can be favorable to reduce subsequent fatigue crack growth. If a sufficient plastic zone develops with compressive residual stresses at the crack front, the resulting crack closure can cause fatigue retardation. A cell type crack mesh was used to examine the hydrostatic test with crack growth and crack face closure. Figure 2 shows an example of the cell type crack mesh for comparison to the focused type crack mesh in Figure 1. Using elastic-plastic FEA, the test pressure is applied to develop a crack front plastic zone, and subsequent crack growth used the node release method in Warp3D. This method allows all crack front nodes to have the symmetry constraint released gradually to advance the crack front.

Typically, several thousand time steps are used for a node release analysis to model the cyclic loading and for the node releases at specified time steps. Results indicate that a test pressure giving a hoop stress of 95% of the specified minimum yield strength gives the most benefit to crack growth retardation. Higher hydrostatic test pressures reduce the overall effect of the compressive residual stress zone, which is a counter-intuitive result.

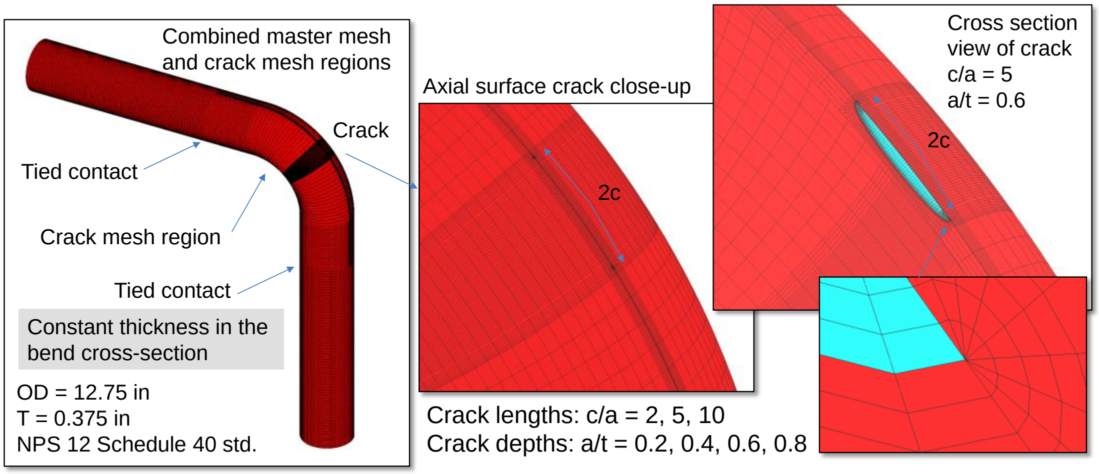

Fig.1 – Piping bend elbow with surface crack focused type mesh and using tied contact. Used 12 crack meshes to obtain initial limting flaw curve using ductile tearing analysis.

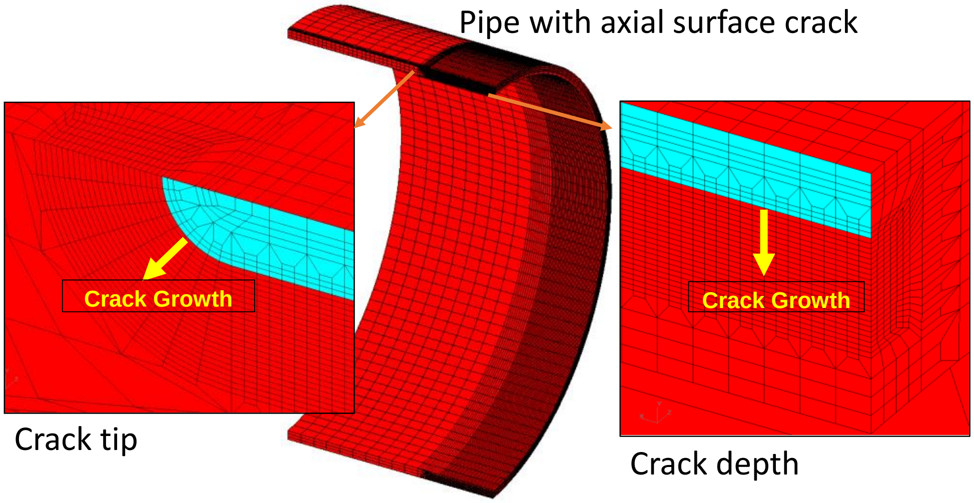

Fig.2 –Pipe with axial surface crack cell type mesh to model crack growth and crack face closure.

3. Conclusions

Two examples were presented investigating the details of using 3-D crack meshes in Warp3D FEA for FFS assessments. A focused type crack mesh was used for ductile tearing analysis to determine if a cylinder model can be used as approximation for a pipe elbow. The limiting flaw curve comparison shows an elbow model is needed. A cell type crack mesh was used to examine pipeline hydrotest pressures to cause fatigue crack retardation by crack face closure. The test pressure giving the most benefit was determined.

Acknowledgements

The first case study is from an NAFEMS conference paper with co-author Erik Jensen.