

Friction stir plug weld crack meshing for NASA

G. Thorwald, Ph.D.

Quest Integrity Group, Boulder, Colorado, USA

Abstract: Friction stir welding is used to fabricate the NASA Ares I rocket upper stage fuel tank. The friction stir plug weld fills the hole left when the stir weld tool is removed. NASA engineers want to examine fracture conditions in the plug weld due to possible imperfections to assure safety and reliability. An automated and parametric crack mesh generator is needed to quickly and easily examine numerous cases. The plug weld crack meshing approach and its reliance on Abaqus for analysis is discussed, and crack mesh basics are reviewed. The choice of crack shape and 39 additional geometric and mesh control parameters, such as the plug diameter, transverse weld sizes, and plug bevel angle affect how the crack mesh is created. Using the plug bevel angle option or surface crack choice requires tied contact in Abaqus to connect some of the mesh zones. Using the through-thickness crack shape permits results comparison of tied contact versus merging coincident nodes to show that the crack front J-integral computed by Abaqus is unchanged, which gives confidence for cases where the tied contact must be used. The crack follows the plug circumference, which creates the possibility of crack face closure and the need for general contact modeling in Abaqus. The crack mesh examples used show the variety of possible geometries in the mesh zones between the plug bevel surface and the surrounding transverse welds.

Keywords: crack mesh, fracture mechanics, friction stir weld, friction stir plug weld, general contact, J-integral, mixed-mode opening, NASA Ares I, stress intensity, tied contact.

1. Introduction

The Ares I rocket upper stage fuel tank is fabricated using several longitudinal, circumferential, and gore section friction stir welds to join the Aluminum-Lithium (Al-Li) alloy panels forming the cylinder drum and vessel heads (Davis, 2008), (Ding, 2008).

Friction stir welding uses a rotating pin tool to heat the metal by friction, and stir the solid plasticized metal to join each component at the interface. Friction stir welding produces a high quality and high strength weld. Since the metal remains in the solid state (no melting) compared to fusion welding (molten metal), several possible defects are avoided, such as: liquidation cracking, solidification cracking, oxide formation, and thermal distortion. In addition, the friction stir weld retains material properties close to the parent metal (Ding, 2006).

For the axial welds, the stir weld tool can travel into an excess material region at the end of the panel, which is later trimmed off and removes any hole left by the stir weld tool. When the circumferential stir weld around the cylinder drum is finished, a hole is left in the metal when the stir weld tool is removed. A friction stir plug weld is used to fill the hole and complete the circumferential weld. Figure 1 shows a picture of a friction stir plug weld in a test panel, along with the weld termination hole and a hole prepared for the plug (NASA, 2008). The weld termination hole is left in the panel when the stir tool is removed. After the hole is prepared, the

plug is stir welded into the plate, and is larger than the weld in this case. Notice the stir weld affected region in the material around the plug.



Figure 1. The test panel shows an example of a friction stir weld and a friction stir plug weld that fills the hole that is left when the stir weld tool is removed (NASA, 2008).

The final friction stir plug weld is the geometry considered to create the parametric and automatic crack mesh generation for finite element analysis (FEA) using Abaqus. The plug weld crack mesh geometry is a rectangular plate with the circular plug in the center. Two concentric circular zones surround the plug, which allows modeling of the stir weld heat-affected zones (HAZ, a terminology borrowed from traditional welding) between the plug and the transverse weld zones, and permits adjustment of the radial location of the crack. The crack is located at the boundary between the two circular HAZ and can be located at any angle around the plug. There are seven transverse weld zones to allow various possibilities of the plug being within the center weld, or extending further out into the transverse HAZ. Figure 2 shows an example of the plug weld mesh with color-coded material zones. The base plate materials are at each end of the rectangular plate model, for a total of 12 mesh and material zones. Figure 3 shows a close-up of the center plug region and an example of the crack location on the circular arc around the plug between the blue and yellow mesh regions.

A rectangular plate is used since the plug size is small relative to the fuel tank curvature and for modeling test panels to compare results to physical tests. The 12 mesh zones help support a large variety of plug weld geometry cases by adjusting their sizes, and mesh refinement, and by assigning different or the same material properties. For example, if the five center transverse zones are given the same material properties, then the plug is entirely within the transverse weld metal. In contrast, by assigning the same material to the center three transverse weld zones, the plug extends out to the transverse weld HAZ.

The result of this project is a parametric and automatic crack mesh generator to allow quick and easy crack mesh generation for a wide variety of plug weld cases. Quick crack mesh generation allows examination of many cases, and helps improve the safety and reliability of the friction stir plug weld. The plug weld crack mesh is an addition to the FEACrack™ software (Quest Integrity, 2009), which creates Abaqus™ input files for analysis.

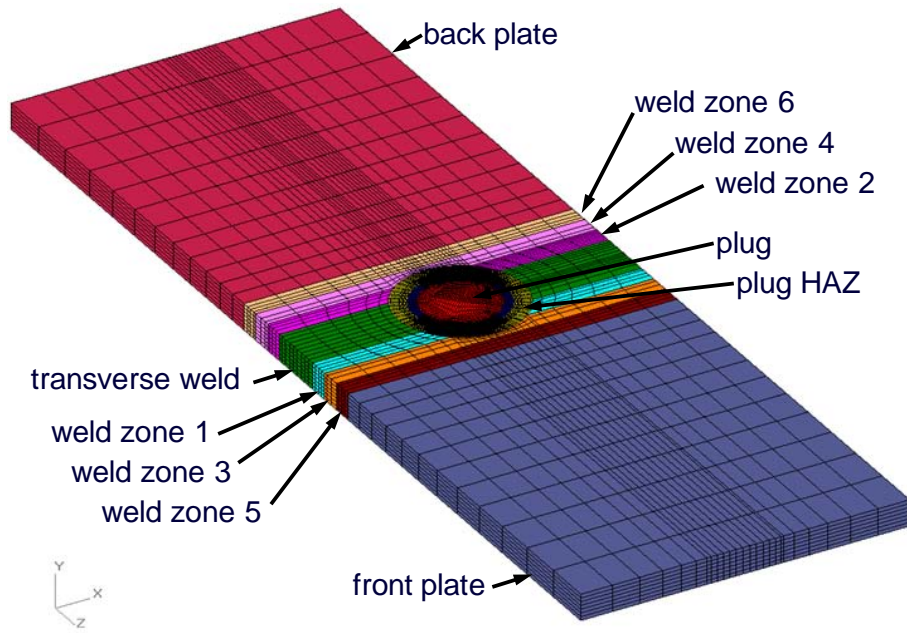


Figure 2. The friction stir plug weld crack mesh is shown with 12 color-coded material zones to support a variety of plug weld cases.

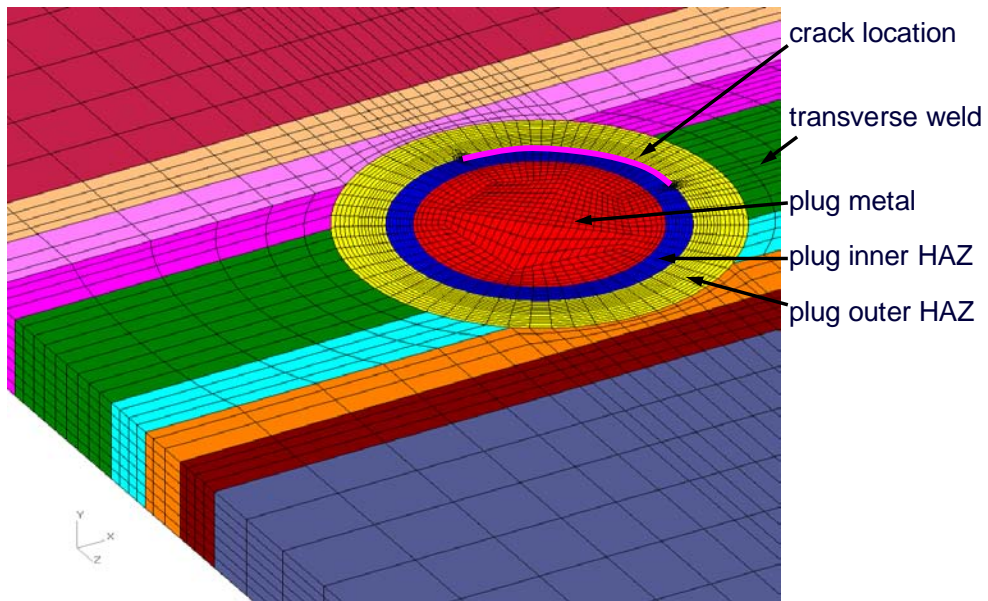


Figure 3. The crack is located on the circular arc between the inner and outer plug heat affected zones, between the blue and yellow regions in this close-up of the plug mesh.

1.1 Crack mesh basics

A crack mesh is used to model a sharp defect in a structure to compute the crack front J-integral (an energy release rate) and stress intensity K (a fracture mechanics parameter) values for the given crack size, structural geometry, loading, and material properties (Anderson, 2005). Computing J-integral and K values by FEA is very useful when they are not available from an existing source. The J-integral and K crack front values are used in fracture mechanics evaluations to determine critical crack sizes for fracture, and to compute crack growth rates for fatigue and stress corrosion crack growth.

Figure 4 shows the basic features of a 3D surface crack mesh. The focused mesh pattern at the crack front uses brick elements, which are required to compute the J-integral using the Abaqus contour integral command (Abaqus, 2009). The wedge-shape elements at the crack front are actually collapsed brick elements and help model the crack front stress singularity. In this case there is a symmetry plane through the crack to better show the crack face and crack front. High mesh refinement along and around the crack front is typical.

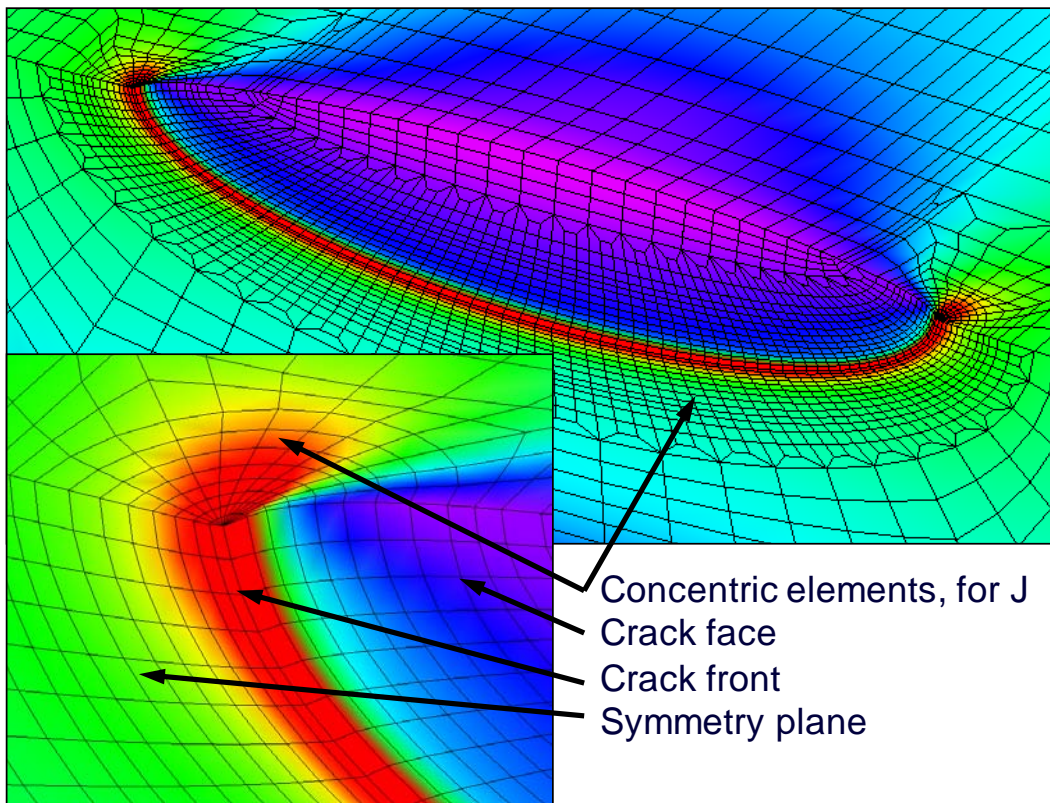


Figure 4. The basic features of a 3D crack mesh include the focused mesh pattern using concentric rings of brick elements along the crack front to compute the J-integral.

Generating 3D crack meshes can be challenging due to several meshing and analysis requirements. For 3D crack meshes in general, and the plug weld in particular, the challenges include:

- Case studies with many crack sizes, requiring a new crack mesh for each case
- Choice of several crack shapes: through-wall, surface, 360° partial-depth, and partial-arc
- Crack aspect ratios: small to large cracks, and shallow to deep cracks, which require changes to the crack mesh for each size
- Variety of geometry dimensions and options: the plug, transverse welds, and plates
- Cracks located anywhere around the plug on a curved and beveled surface
- High mesh refinement at the crack front
- The focused mesh pattern with brick elements at the crack front required for the J-integral
- Sorted crack front node set data for the input file to compute the J-integral
- Post processing to extract and tabulate crack front J and K results for each case

A single crack meshing challenge is perhaps not overwhelming, but all the meshing challenges must be met for every crack mesh. The sum of the meshing challenges can make 3D crack meshing a difficult and time-consuming task (Thorwald, 2003). An automatic and parametric crack mesh generation software can simplify these tasks and allow many analyses.

2. Plug weld mesh

The 39 friction stir plug weld mesh parameters include the plate, plug, and transverse weld zone dimensions, the plug bevel options, and controls for the mesh refinement in each mesh zone. These parameters are in addition to the crack shape choice, the crack dimensions, and the location of the crack around the plug circumference. To generate the plug crack mesh, the mesh zones help divide the model into several simpler steps for programming the parametric mesh generation routines. The crack mesh generation starts with the rectangular crack mesh primitive, which is then geometrically transformed to a ring shape. The crack mesh primitive contains the crack and the focused mesh along the crack front. The inner plug mesh is added in the center of the ring, and possibly transformed again if a plug bevel option is active. Then the transverse weld and plate mesh zones are added around the plug to complete the mesh. Figure 5 shows the first three steps: crack mesh primitive, the transformed ring shape, and adding the center plug mesh.

This method of starting with the rectangular crack mesh primitive and a geometric transformation has proven to be very adaptable and supports locating 3D crack meshes in a variety of geometric shapes. In the plug weld, the crack mesh primitive is generated for the chosen crack shape, such as a surface, through-wall, 360° partial-depth, or partial-arc crack.

In some cases when the center plug and transverse weld zones are added, the nodes at the interface between the crack mesh and neighboring mesh zones can be aligned and merged to join the mesh zones. For some of the model options, such as a surface crack or outer plug bevel, all the interface nodes cannot be aligned so tied contact is used to join the mesh zones.

A full crack is used in the mesh without any symmetry planes to support a crack located anywhere on the circular arc around the plug. When using a full crack in the mesh, the crack faces are not

visible from the exterior of the mesh. Only the focused mesh pattern at the crack tips can be seen on the top surface of the mesh. From the FEA results, the deformed mesh can be used to exaggerate the crack opening and help view the crack faces and crack front.

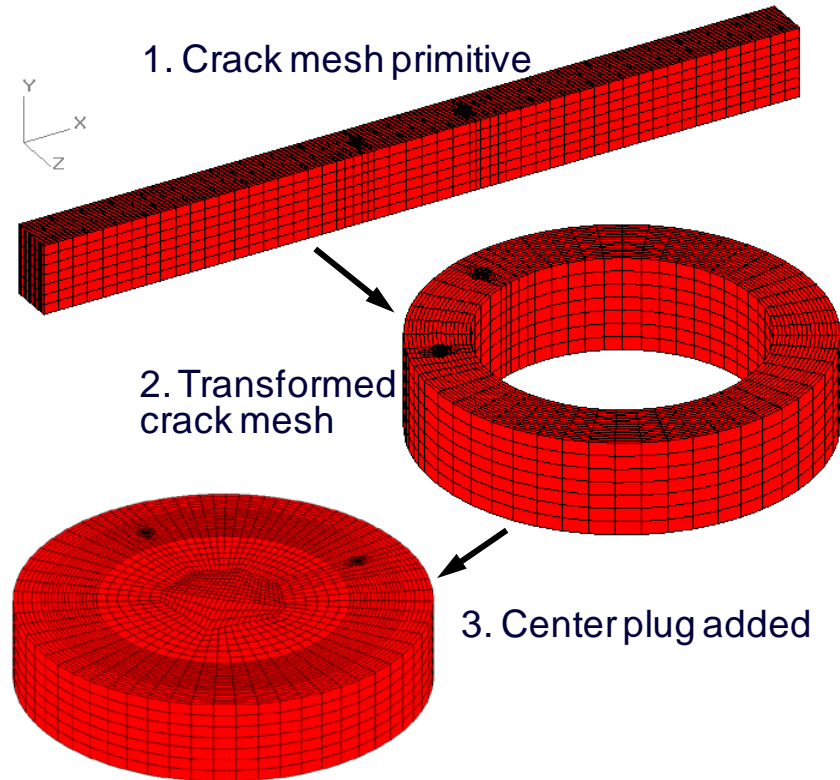


Figure 5. The crack mesh begins as a rectangular primitive (1) and is then geometrically transformed into the ring shape (2); the plug mesh is added to fill the center zone (3).

2.1 Plug bevel option

The plug weld mesh has an inner bevel and outer bevel option. The simplest case for the transverse mesh zones is when there is no outer plug bevel, which gives an outer cylinder profile through the plate thickness. Figure 6 shows the transformation regions used for the inner plug bevel option. Five regions are used to preserve the crack front element shapes. In Figure 6, the top and bottom view of the plug mesh zones show the smaller bottom plug diameter (yellow elements) and how the outer plug HAZ (red elements) is stretched to fill the volume below the plug.

Figure 7 shows a cut-away view of the plug mesh to reveal how the inner plug bevel option retains the outer plug cylinder profile through the plate thickness, but allows the inner plug HAZ to have a bevel angle, which adjusts the angle of the crack face. The bevel puts the crack face on a conic surface, defined by the plug top and bottom diameters. The inner plug bevel option still allows node merging to join the mesh zones.

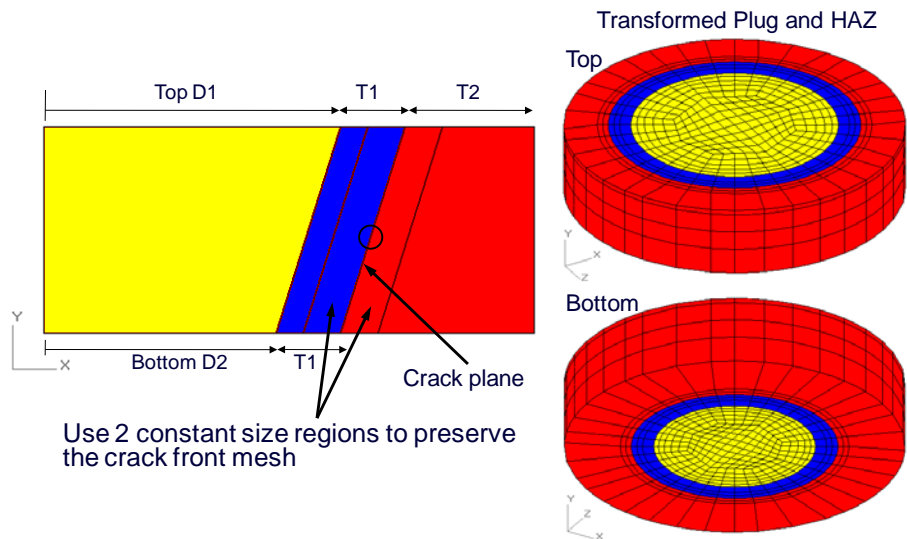


Figure 6. The inner plug bevel option uses five regions to preserve the crack front element shapes in the inner bevel geometry transformation.

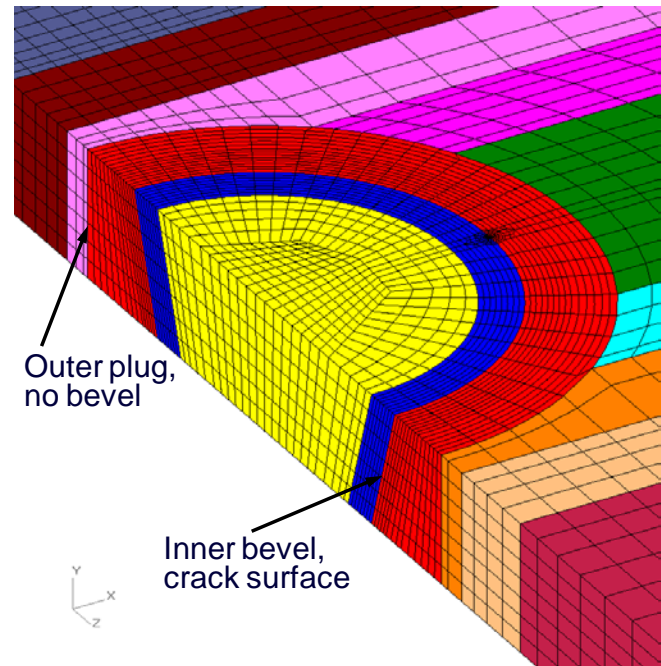


Figure 7. The cut-away mesh view shows the inner plug bevel and the outer plug cylinder profile through the plate thickness.

The outer plug bevel option is a more general case with the bottom plug outer diameter also included in the plug transformation, which gives the outer plug a conic surface. The outer plug bevel has three cases depending on the bottom plug diameter relative to the transverse weld zone sizes:

- A. Small bevel: the bottom of the plug is within the outer transverse weld zones
- B. Medium bevel: the bottom of the plug is within the transverse weld HAZ
- C. Large bevel: the bottom of the plug is within the center transverse weld zone

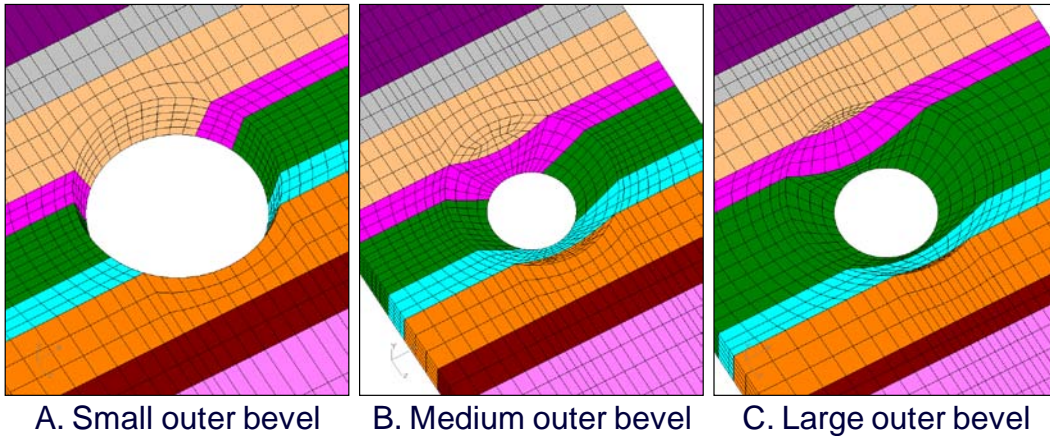


Figure 8. The plug has been removed to show the 3 plug outer bevel cases; the transverse mesh zones have been adjusted and elements added to follow the plug's outer conic surface.

The plug has been removed from the meshes in Figure 8 to better show the transverse weld zones. Figure 8-A shows that for the small outer bevel the transverse mesh zones can be adjusted around the plug to follow the plug's conic surface. At the intersection of the plug's outer conic surface and the transverse weld planar surfaces, the edge curves are hyperbolas. The hyperbola curves vary for each case and are defined by the location of the plane and conic surface intersection, which depends on the relative sizes of the plug and transverse welds. Adjusting the transverse mesh zones is preferred in this case rather than adding more elements to avoid very thin brick elements, which would be unfavorable for the analysis.

Figure 8-B shows the medium bevel case and the extra elements added in the transverse weld zones to fill the volume below the plug's conic surface. The volumes in each mesh zone below the plug conic surface have an interesting profile since they must follow the transverse mesh zone planes to preserve the material property regions and match the conic surface of the outer plug. The added elements are aligned with the existing transverse weld zones and the nodes are merged to join the new and existing elements.

Figure 8-C shows the large bevel case. The added elements below the outer plug's conic surface in the center transverse weld zone now completely surround the bottom of the plug. The nodes in the added elements are again aligned and merged with the neighbor transverse weld zones.

Since the mesh pattern for the outer plug conic surface and the matching surface in the transverse weld zones cannot be generally aligned, tied contact is used to join the plug and transverse weld mesh zones for the outer plug bevel option.

Tied contact

For some of the crack shape choices, such as the through-wall and 360° partial-depth cracks, the nodes at the interface between the outer plug and transverse welds can be aligned and the nodes merged to join the mesh zones. But when the plug model uses the outer bevel option or a surface crack, the interface nodes cannot generally be aligned, so tied contact is used to join the mesh zones. Tied contact is a mathematical constraint method that joins two dissimilar mesh surfaces, which is crucial for these cases in the plug weld mesh. It is fortunate that Abaqus has a very good and easy to use tied contact option (Abaqus, 2009).

Nodes merged between mesh zones: 360° crack

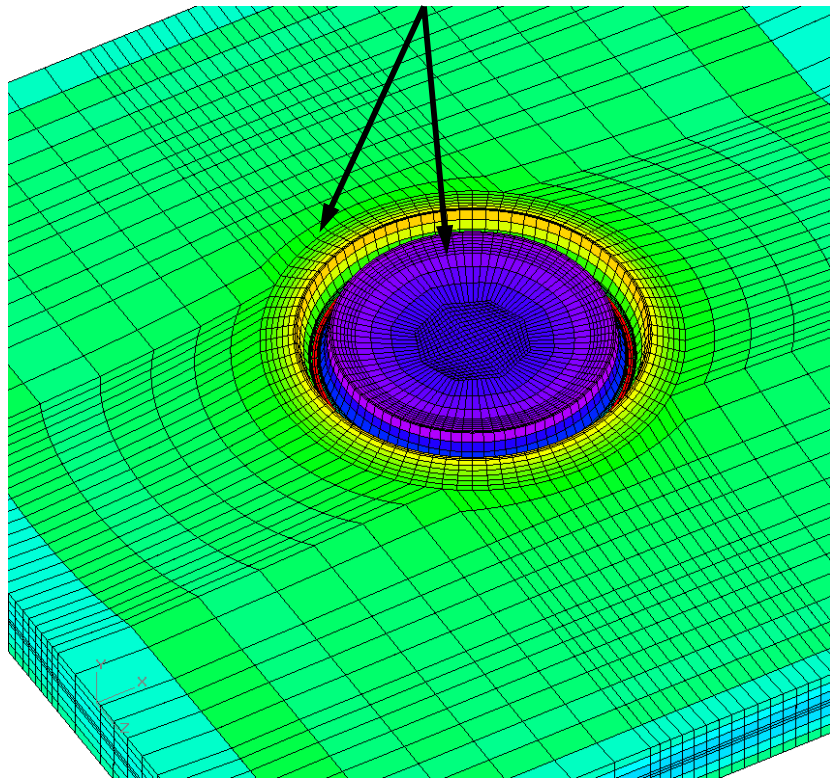


Figure 9. The 360° partial-depth crack has nodes merged to join the mesh zones; the mesh has an exaggerated deformed shape to show the crack opening and Von Mises stress results.

Figure 9 shows a close-up of a 360° partial-depth crack example, which has the nodes aligned between all the crack, plug, and transverse weld mesh zones, and then merged to join all the mesh zone elements. The deformed shape is exaggerated, and the Von Mises stress results (a stress

invariant value) give the color map on the mesh. The red elements at the crack front show the high crack front stress, while the purple elements in the plug center show low stress, since the top of the plug is out of the load path due to the crack shape. The J-integral is computed by Abaqus at all the crack front nodes.

Figure 10 shows a close-up of a surface crack example using tied contact to join the mesh zones. The number and size of elements between the surface crack zone, the inner plug, and transverse weld zones are different in general, so tied contact is used to join the mesh zones. The exaggerated deformed shape shows the surface crack opening. The results show high Von Mises stress (red elements) at each crack tip. Behind each crack face is a low stress region (blue and purple elements) for the elements out of the main load path due to the crack shape.

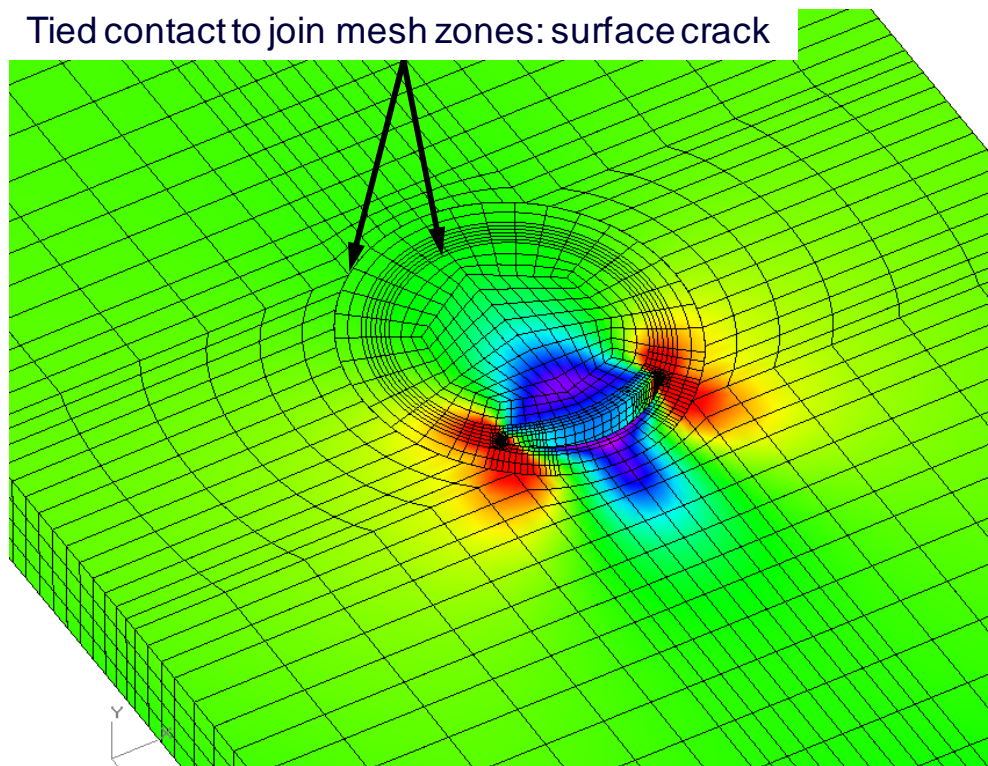


Figure 10. The surface crack example uses tied contact to join the mesh zones; the close-up of the deformed plug mesh shows the dissimilar element sizes between the mesh zones.

When the nodes between the plug and transverse mesh zones are aligned and can be merged, there is an option to activate the tied contact instead, so that results using merged nodes or tied contact can be compared. Figure 11 shows a comparison of the crack front normalized J-integral results from two analysis cases for a through-wall crack using merged nodes and using tied contact. The normalized distance along the crack front through the plate thickness is d/T (d is the crack node position, and T is the plate thickness). The normalized J-integral result values agree to the fifth decimal place, which demonstrates that with sufficient mesh refinement, the tied contact does not

adversely affect the crack front results. This result gives confidence that the required use of tied contact for some cases will not affect the crack front J results.

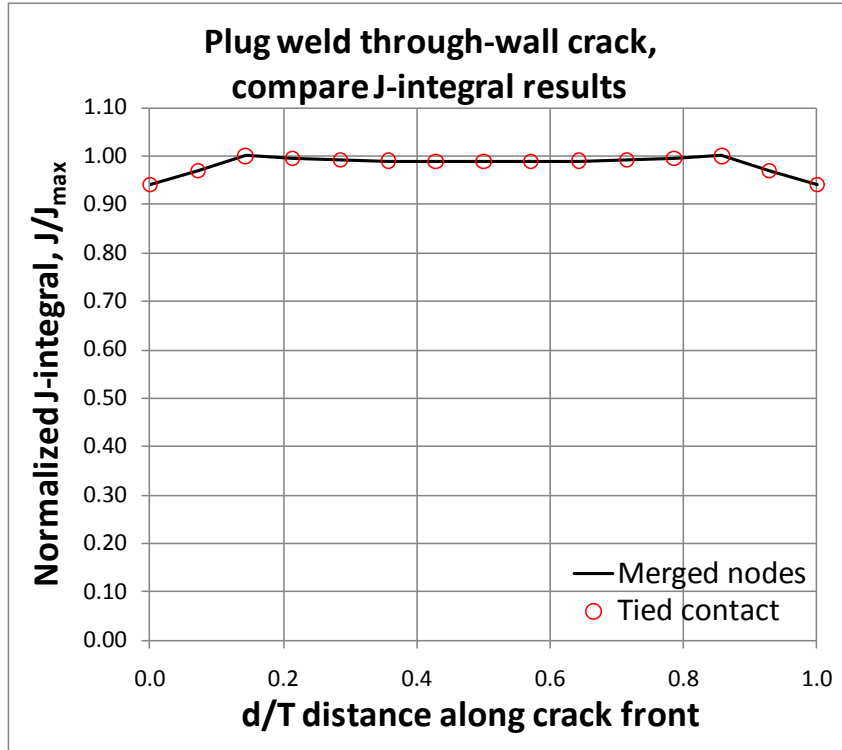


Figure 11. The through-wall crack normalized J-integral results agree using merged nodes or tied contact to join the plug weld mesh zones.

Figure 12 shows an overhead view of the through-wall crack mesh used to compare the crack front J-integral results. The crack is located on the plate center line so that the J-integral results are the same for both the right and left crack fronts.

Crack face contact

Depending on the location of the crack around the plug and the loading applied to the plate, crack closure is possible. The crack front opens in a region with tensile stress, and the crack front can close due to a region with compressive stress. An element-based surface is defined on each crack face so that general contact can be included in the analysis (Abaqus, 2009). When crack face contact is included, additional solution iterations are needed, and the run-time will be longer. Often an initial analysis without the crack face contact data can be used to determine if crack closure is likely, so that the contact data is included only when necessary.

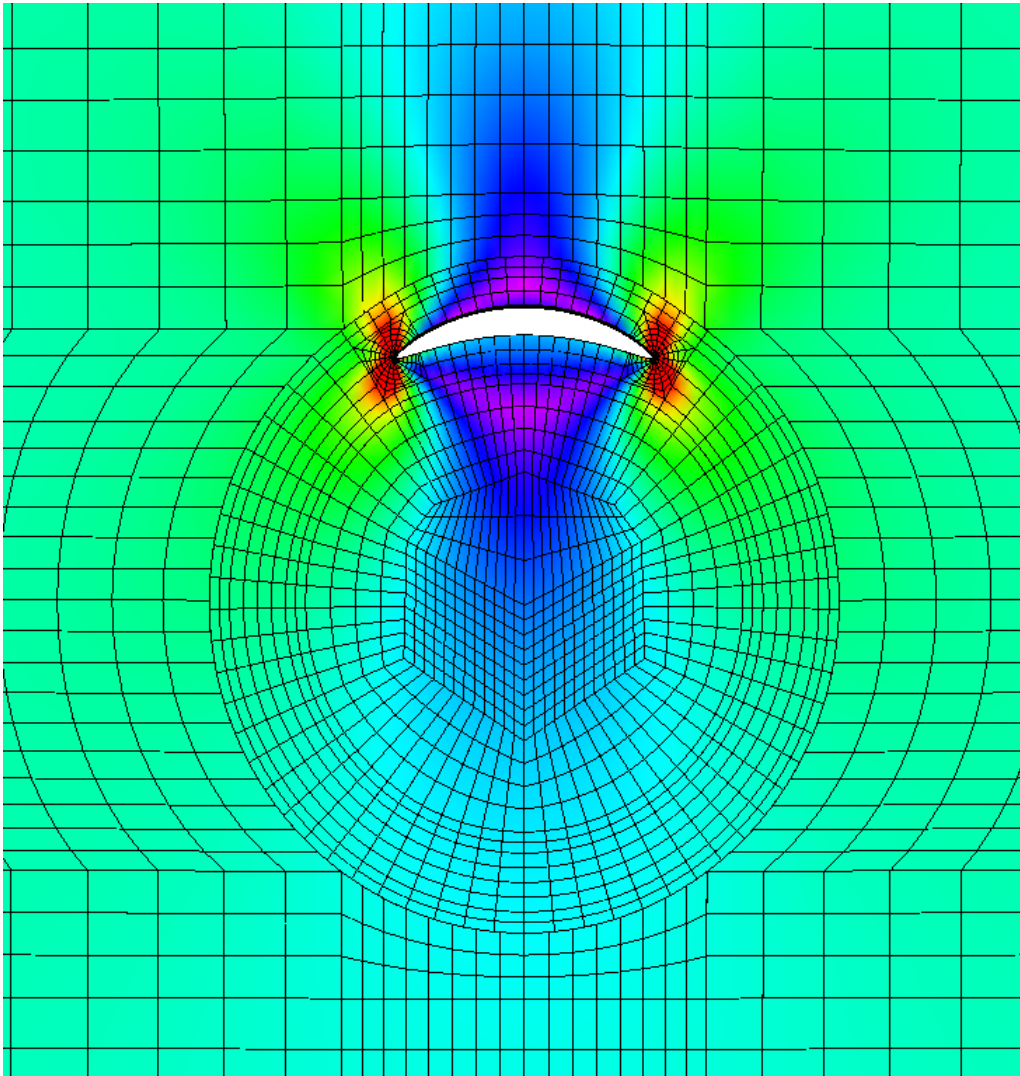


Figure 12. An overhead view of the through-wall crack mesh used to compare J-integral results using merged nodes and tied contact; the deformed shape shows the crack opening.

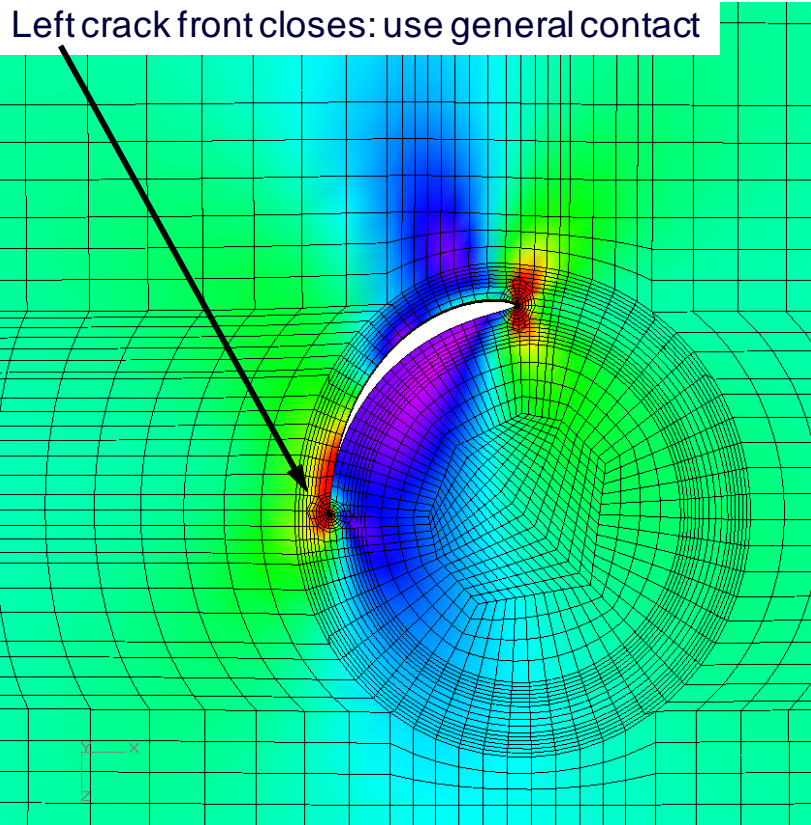


Figure 13. The through-wall crack example has a 90° arc, which causes the left crack front to close due to the tensile loading applied to the plate; contact data is used to model the crack surface closure.

Figure 13 shows an overhead view of a through-wall crack example, which has a 90° arc around the side of the plug. The crack location is used to show crack opening at the right crack front, and crack closure at the left crack front. The exaggerated deformed shape shows the crack opening gap between the crack faces along most of the crack front, and the partial crack closure near the left crack front where the crack faces are in contact.

Figure 14 shows the mixed-mode stress intensity K results for the two crack fronts. The mixed-mode K values are computed using the crack opening displacements to get each K component. The six curves correspond to the three opening modes for the two crack fronts. The right side crack front curves have solid symbols, and the left side crack front curves have open symbols. The symbol shape is the same for each mode; for example, circle symbols are used for mode I. Crack mode I is the opening mode where the crack faces separate. Mode II is in-plane shear where the crack faces are sliding relative to each other, and mode III is the out-of-plane shear where the crack faces move vertically and opposite of each other. The right side crack front has primarily mode I opening, with the largest K_I values (solid black line with solid circle symbols).

The left side crack front has primarily mode II and mode III due to the crack face closure; the left side crack front K_I values are close to zero (solid purple line with open circle symbols) due to the closure.

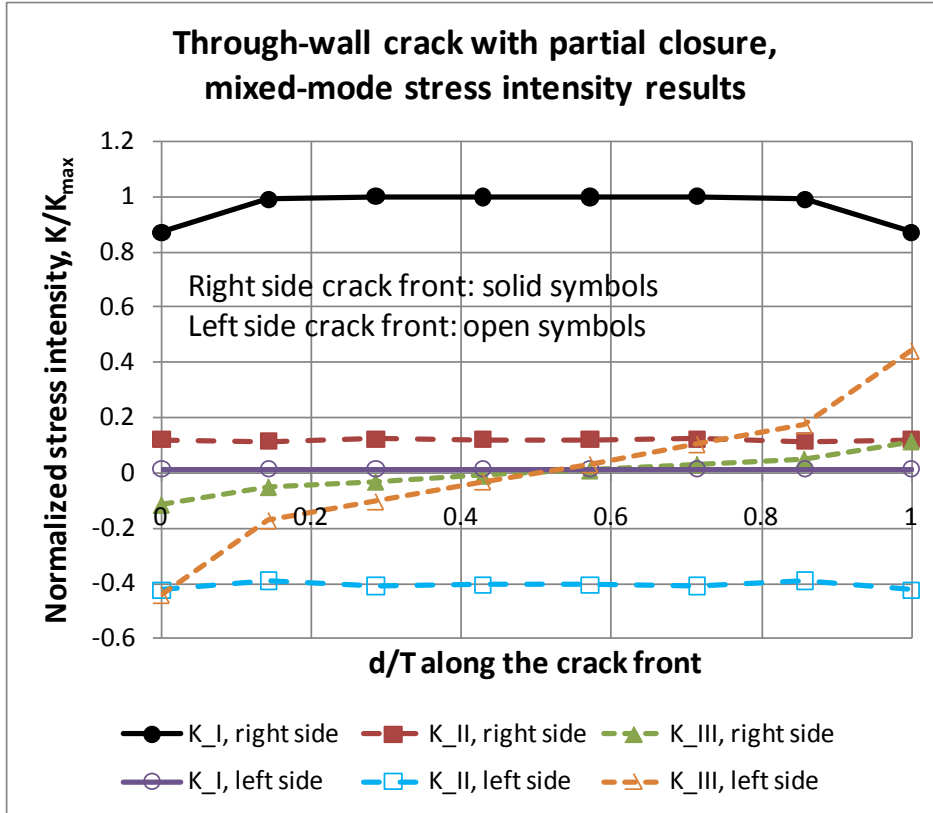


Figure 14. The right side crack front (solid symbols) has primarily mode I opening (black curve), but the left side crack front has mode II (in-plane shear, blue curve) and mode III (out-of-plane shear, orange curve) opening due to the partial crack closure.

3. Summary

A parametric and automatic friction stir plug weld crack mesh generator was developed for NASA, and it provides quick and easy 3D crack mesh creation for finite element analysis to compute crack front J-integral and stress intensity K values. After reviewing some crack mesh basic features, examples and results of the plug weld crack mesh were shown for some of the crack shape and plug bevel options. Tied contact is used to connect the plug to the transverse weld mesh zones when the mesh patterns are dissimilar and the use of tied contact was shown to not affect the crack front results. When crack closure is a possibility due to the crack location around the plug or the applied loading, general contact is used between the two crack faces.

4. References

1. Abaqus Analysis User's Manual, section 11.4.2, "Contour integral evaluation," section 30.3.1, "Mesh tie constraints," and section 31.3, "Defining contact pairs," version 6.9, Dassault Systèmes Simulia Corp., Providence, RI, USA, 2009.
2. Anderson, T. L., "Fracture Mechanics, Fundamentals and Applications," 3rd Ed., CRC Press, 2005.
3. Davis, D. J., and C. J. McArthur, "NASA Ares I Crew Launch Vehicle Upper Stage Overview," NASA Technical Reports Server (NTRS), <http://ntrs.nasa.gov>, document 20090004512_2008048207.pdf, p. 6, 2008.
4. Ding, J., B. Carter, K. Lawless, A. Nunes, C. Russell, M. Suites, and J. Schneider, "A Decade of Friction Stir Welding R&D at NASA's Marshall Space Flight Center and a Glance Into the Future," NASA Technical Reports Server (NTRS), <http://ntrs.nasa.gov>, document 20080009619_2008009118.pdf, 2006.
5. Ding, J. "NASA's MSFC Welding Development For Ares I," NASA Technical Reports Server, <http://ntrs.nasa.gov>, document 20090015019_2009013964.pdf, 2008.
6. Quest Integrity Group USA LLC, "FEACrack user's manual," version 3.1, Boulder, Colorado, 2009.
7. NASA Marshall Spaceflight Center, plug weld test panel picture, "FEACrack friction stir plug weld," project proposal, 2008.
8. Thorwald, G., "3-D Crack Analysis Challenges," West Regional Abaqus Users' Meeting, October 21-22, 2003.

5. Acknowledgment

This project was generously sponsored by the NASA Marshall Spaceflight Center (MSFC), Huntsville, Alabama, and administered through the Jacobs ESTS Group. Thank you to Doug Wells, Jonathan Burkholder, and Phillip Allen at NASA MSFC, and Dennis Lambert at Jacobs ESTS Group for their help defining the friction stir plug weld project and reviewing the new software features, and for the permission to use the plug weld test panel picture in this paper. Thank you to Daniel Morris and Dan Revelle at Quest Integrity Group for the plug weld input controls added to the FEACrack graphical user interface.