

Linear Elastic Fracture Mechanics (LEFM) Analysis of Flaws within Residual Stress Fields

David Woyak¹, Brian Baillargeon, Ramesh Marrey, and Randy Grishaber²

¹Dassault Systemés SIMULIA Corporation &

²Cordis Corporation, a Johnson & Johnson company or its affiliate Conor Medsystems, LLC

Abstract: A procedure for calculating Linear Elastic Fracture Mechanics (LEFM) Stress Intensity Factors associated with postulated flaws located within regions of residual stress that result from large strain forming processes is presented. The procedure is validated by analyzing a structural system with residual stresses representative of a plane strain 3-point bending configuration, for which a known solution exists. The procedure is also applicable to the analysis of flaws in residual stress fields that result from in-service loads leading to large plastic strains, as illustrated by the determination of stress intensity factors associated with a postulated flaw in a deployed coronary stent. The procedure utilizes the ABAQUS import capability where the residual stresses are imported along with the updated model geometry. The material is redefined upon import to be linear elastic and the fracture analysis is performed by releasing the flaw face constraints that were active during the forming or otherwise large strain analysis.

Keywords: Coronary Stent, Import, Large Strain, Fracture Mechanics, Stress Intensity Factor

1. Introduction

In general, metal forming processes are highly nonlinear in terms of both geometry (large displacements) and material response (plasticity, large strains). Therefore, the post-formed state of a component, in the absence of an anneal treatment, is often characterized by residual stresses that represent a self-equilibrating tension/compression force system within the component. Usually the post-forming residual stresses are sufficiently less than the strain hardened yield stress that the subsequent response due to in-service loading remains linear elastic. The formed component may need to be evaluated for fracture and fatigue crack growth behavior if flaws are present or introduced during the forming process or are introduced post-forming via corrosion, aging or through in-service damage. The influence of the residual stresses should be included when evaluating a formed component's fracture and crack growth characteristics. This paper describes a procedure for calculating Linear Elastic Fracture Mechanics (LEFM) stress intensity factors for postulated flaws located within regions dominated by high tensile residual stresses.

2. Procedure

The procedure for calculating Mode I, II and III stress intensity factors (K_I , K_{II} , K_{III}) in a residual stress field will be referred to in this paper as RS-LEFM. RS-LEFM utilizes the /Standard to

/Standard import capability, with the forming analysis being the parent to the import analysis which is used to perform the fracture mechanics evaluation. The forming analysis is performed in a usual manner except that focused quarter point element meshes are included to represent postulated flaws. Alternatively, the postulated flaws can be introduced into sub-models of a flaw-free forming analysis, and the sub-models then become the parents to subsequent import analyses. The term “quarter point element” in this application refers to a high order quadratic element, such as the C3D20R, where the mid-side nodes for element edges that emanate from the crack front are moved to the $\frac{1}{4}$ edge length position. For linear elastic material response of an opening flaw the quarter point positioning produces the required $1/\sqrt{r}$ strain singularity.

During the parent forming analysis, the flaw faces are forced to remain closed with either point to point constraints or through the use of a contact pair. Point to point constraint methods that are automatically activated upon import and which cannot subsequently be removed, such as a rigid body constraint, are not applicable to this procedure. When using a contact pair, the slave surface should be a node based surface and should not include the nodes located on the crack front. Using a node based slave surface prevents the underlying quadratic elements from being transformed to variable node elements, which would activate an additional mid-face node. The flaw face contact pair surface behavior is defined with “NO SEPARATION” and “ROUGH” friction active so as to tie the slave nodes to the flaw master surface. Additional constraints need to be applied to nodes along the crack front if the focused quarter point element mesh utilized collapsed element faces. At each nodal position along the crack front the multiple nodes from collapsed faces must be constrained to move as one. Defining a master reference node at each position can be used with techniques such as constraint equations, kinematic coupling or connector elements to apply the crack front constraints. Figure 1 illustrates the forming analysis flaw constraint requirements.

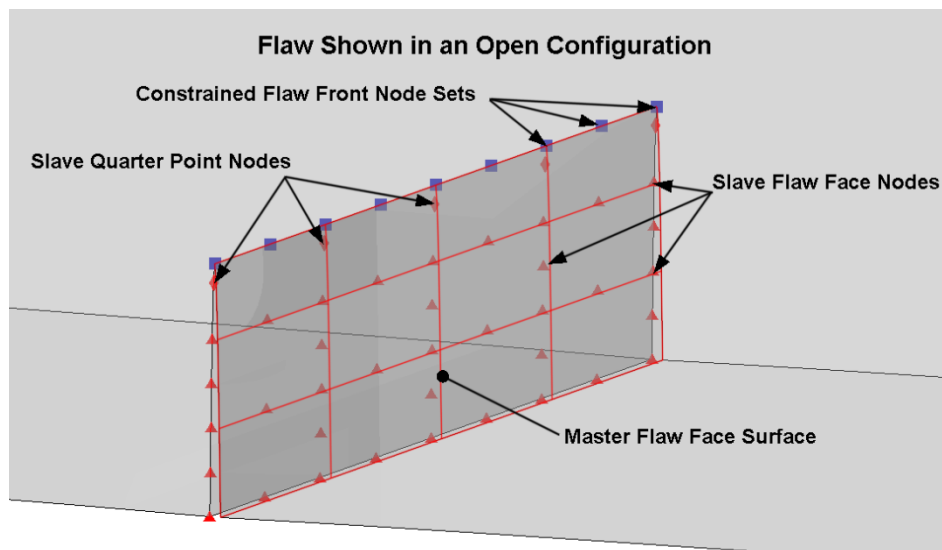


Figure 1: Flaw constraint requirements of the parent forming analysis.

Constraining the flaw to a closed condition during the simulation of the forming process will effectively remove the quarter point strain singularity, thereby allowing the elements of the focused mesh to respond in a fashion that is consistent with no flaw being present. The focused mesh region can thereby adequately represent the elastic-plastic strains and subsequent residual stresses that occur as a result of the forming process. The LEFM evaluation of a postulated flaw is performed by means of the Abaqus “import” procedure for transferring results. The key factors associated with the RS-LEFM import analysis are:

1. The material state, as characterized by the element integration point stresses, is imported by utilizing the STATE=YES parameter on the *IMPORT keyword option.
2. The reference configuration is updated to be the imported configuration by setting UPDATE=YES on the *IMPORT keyword. As a result, the displacements and strains generated in the import analysis are the total values relative to the model’s imported configuration. In effect, the displacements and strains are reset to zero and the new initial geometry is represented by the displaced shape obtained from the forming analysis.
3. The material properties must be redefined with the *MATERIAL and *ELASTIC keyword options so as to remove the plasticity model that was active in the parent forming analysis. The import analysis response is purely linear elastic, which allows for the calculation of stress intensity factors via the Abaqus contour integral procedure.
4. The data lines for the *CONTOUR INTEGRAL, TYPE=K FACTORS keyword option that define the crack surface normal direction or virtual crack extension direction must be relative to the imported configuration.

Dependent on the nature of the parent model, it may be necessary to use the first step of the import analysis to re-establish the constraints, contact, loads and general conditions present at the end of the forming analysis. At the end of this initialization step, the flaw faces are held closed, usually by means of no separation, rough friction contact. The flaw face constraints are released in the next step by removing the flaw face contact pair via the *MODEL CHANGE keyword option. Removing the contact pair allows the flaw face constraint forces to be ramped to zero over the duration of the step, thereby allowing the flaw to open and the $1/\sqrt{r}$ singularity to become active. Since the $1/\sqrt{r}$ singularity is significantly more dominant than the initial residual stresses in the vicinity of the flaw front (i.e., crack tip), reasonable contour integral estimates for stress intensity factors can be obtained. The validation test that follows indicates the greatest accuracy with the RS_LEFM procedure occurs within the first contour integral that consists of the quarter point elements, and reduces slightly when evaluating expanded contour integrals.

It should be noted that Abaqus/Standard will issue a warning when stress intensity factor calculations are requested in an analysis step that incorporates geometric nonlinearities (NLGEOM=YES). The warning is issued because stress intensity factor errors can result if the final orientation of the flaw is significantly different than that defined by the initial model configuration. The warning is not applicable when utilizing the RS-LEFM procedure because the import analysis in which the flaw is allowed to open, utilizes the updated model geometry as the

initial configuration, and the flaw orientation is defined relative to the imported initial configuration.

3. Validation Test

The validation model for the RS-LEFM procedure is based upon a bar in 3-point bending so that the K_I stress intensity factor can be compared directly to a known solution (Hertzberg, 1996). The validation model concept is shown in Figure 2, and consist of a steel frame upon which is mounted an aluminum bar. A steel wedge is then forced between the frame and the bar to load the test fixture, thereby producing a self equilibrated force system in which the stresses are meant to represent the residual stresses that exist after a metal forming process. The validation model has a thru-thickness flaw introduced at the bottom center of the bar. The flaw faces are held in a closed position during the parent forming analysis. In the import analysis the flaw is allowed to open and the final load between the wedge and bar is determined from contact force data. The analysis estimates for the K_I stress intensity factor can then be compared to the known solution.

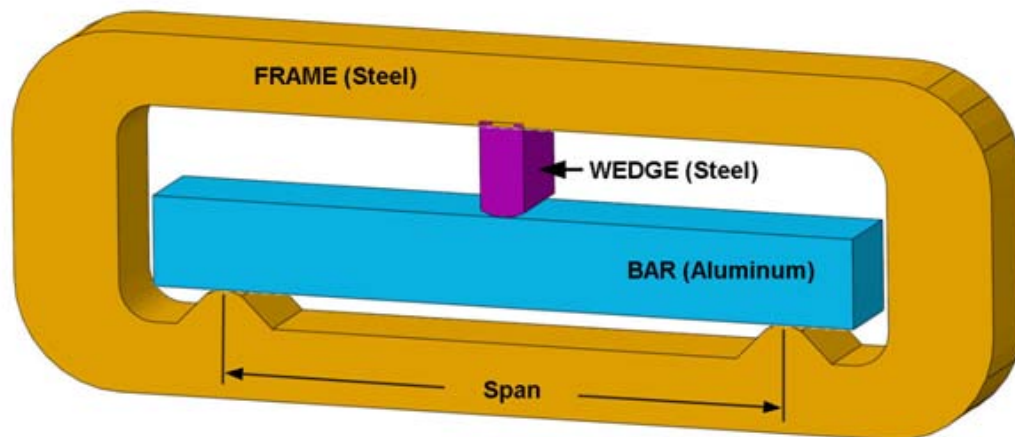


Figure 2. RS-LEFM Validation model concept.

The implemented validation model is somewhat more complicated than the simple rendition illustrated in Figure 2. The implemented model must also illustrate that the load history in terms of prior elastic-plastic response and geometric nonlinear large deflections does not adversely influence the stress intensity factor calculations associated with the post-formed configuration and residual stresses. Figure 3 shows the initial configuration of the validation model at the beginning of the forming analysis. The focused mesh topology at the flaw location is shown in an expanded view. The flaw region is constrained during the forming analysis as described earlier. An initial

gap between the wedge, which is in contact with the bar, and the frame is included to allow for significant transverse expansion of the bar during a forming step in which the length of the bar is compressed from 40mm to 33mm. During this initial forming process plane strain conditions are imposed on all of the parts via the appropriate application of boundary restraints. The steel components are defined as linear elastic, with the aluminum bar having an initial yield stress of 500 MPa and a strain hardened yield point of 700 MPa at 50% plastic strain. The depth of the flaw in this initial configuration is 1mm.

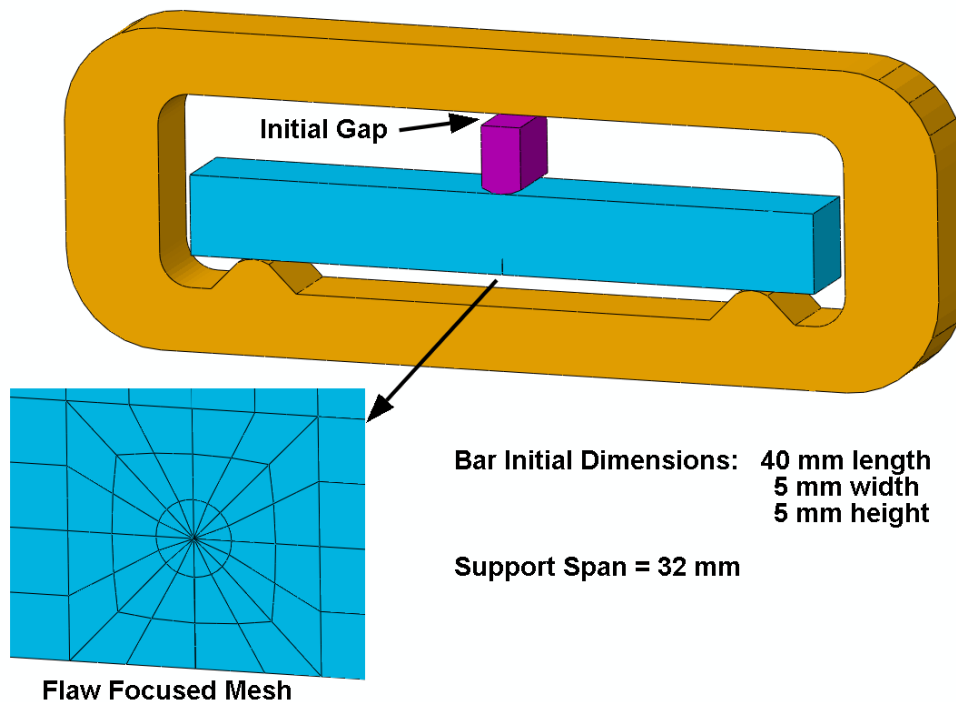


Figure 3: Validation model forming analysis initial configuration.

Figure 4 shows the stress and plastic strain of the bar at the end of the first forming step. Notice that the top surface of the wedge now interferes with the frame, but contact between these two components has not yet been activated. The second forming step removes the compressive loads and allows the system to return to a stress free condition, which also requires the removal of the plane strain constraints on the aluminum bar. The end of the second step, shown in Figure 5, represents a new, stress free configuration, but one in which significant plastic strains exist.

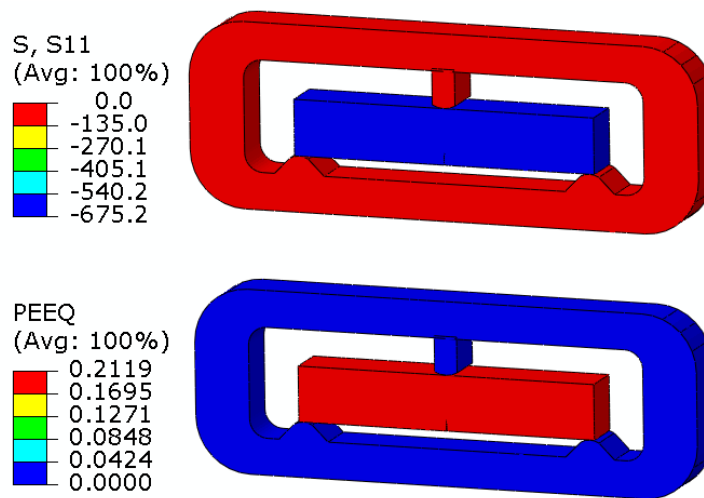


Figure 4: Forming analysis end of Step 1 – large strain axial compression.

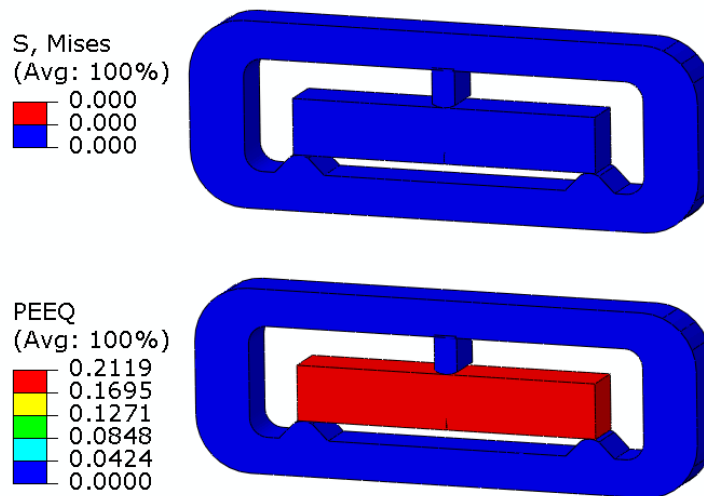


Figure 5: Forming analysis end of Step 2 – new stress free configuration.

At the end of the second step the bar has a length of 33.267 mm, width of 5.008 mm, height of 6.002 mm and a flaw depth of 1.200 mm. The support span for 3-point bending remains at 32mm. In the third forming step the plane strain boundary conditions are re-applied to the bar component by making use of the *BOUNDARY keyword option with the FIXED parameter. Also, the flaw face contact pair used to maintain closure of the flaw remains active and the

interference between the frame and wedge is resolved by activating their contact pair and ramping to zero the allowable interference. The Step 3 stresses correspond to a classical 3-point bending stress field within the bar and associated stresses within the frame and wedge. The model's force system is fully self-equilibrated, since the reaction forces associated with preventing system rigid body motion are identically zero. Figure 6 shows the bending stresses and total bending strains that exist at the end of step 3. Notice that the tensile stresses and total strains at the bottom of the bar are not affected by the presence of the closed flaw in the focused mesh.

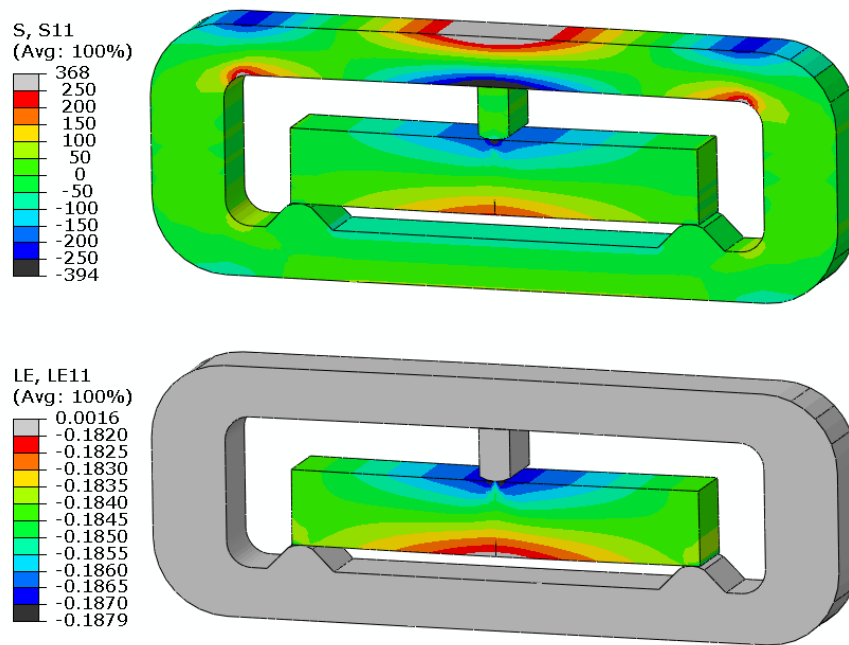


Figure 6: Bending stresses and strains at the end of forming analysis step 3.

Even though the forming steps 1 thru 3 all required geometric nonlinear large deflection to be active, none of them resulted in a significant change in the initial orientation of the flaw. Therefore, in order to verify that the RS-LEFM technique is applicable for cases where the flaw's focused mesh changes orientation during the forming analysis; additional steps were included to impart a 90 degree rigid body rotation to the entire assembly. The final configuration of the forming analysis is shown on the left side of Figure 7, with the bending stresses corresponding to the global 2-direction (S22) rather than the global 1-direction (S11) as was the case in Figure 6. On the right side of Figure 7 is shown the bending stress contour plot obtained from the end of the import analysis fracture evaluation. The influence of the open flaw singularity on the stress distribution is clearly illustrated. The load applied to the bar by the wedge in the open flaw condition was 758.14 Newtons as compared to a value of 819.11 prior to releasing the flaw face contact restraints, representing a 7.4% drop in the force applied to the bar.

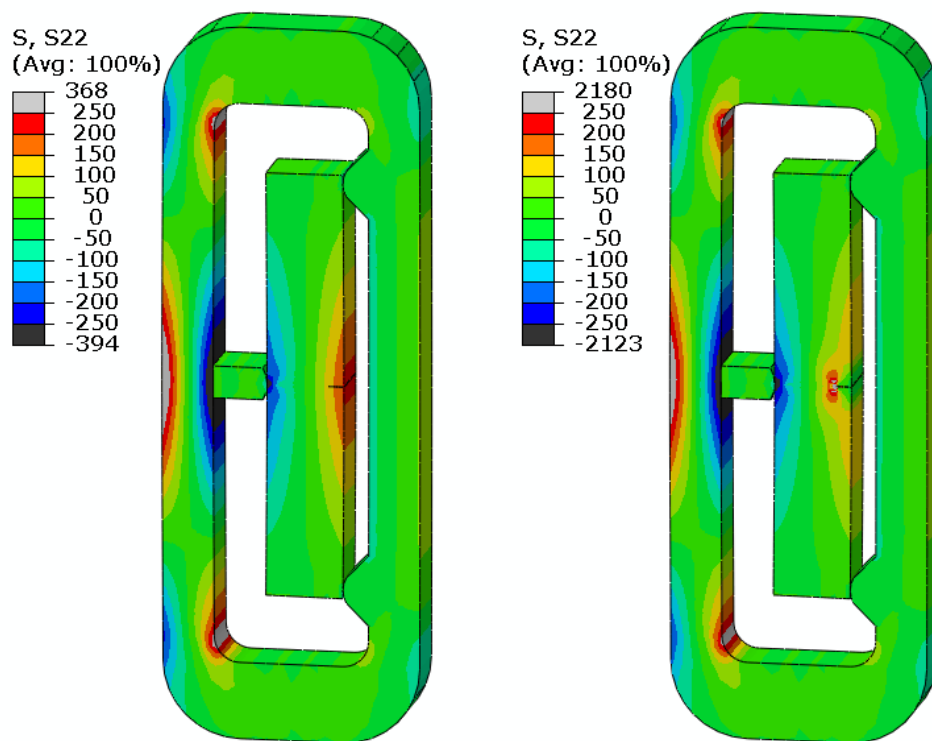


Figure 7: Bending stresses before and after the flaw is allowed to open.

The 3-point bending plane strain solution for the K_I stress intensity factor is $387.0 \text{ MPa} \sqrt{\text{mm}}$, based on a 758.14 Newton load, a 32.0 mm span, and the post-formed bar dimensions of 5.008 mm width, 6.002 mm height and a 1.200 mm flaw size. The Abaqus contour integral estimates, as shown in Table 1, are the most accurate for the first contour containing the quarter point elements, with the prediction being 1.2% below the known solution. The degree of accuracy in the solution is well within acceptable levels for finite element based LEFM evaluations.

Table 1: Abaqus Contour Integral Results

	Contour Integral 1	Contour Integral 2	Contour Integral 3
Stress Intensity Factor	382.4 *	375.5	362.5
% Error	-1.2%	-3.0%	-6.3%

* Corner node positions along the flaw front listed 375.5 and the mid-side node positions listed 391.1, with the average value along the flaw front being 382.4.

4. Practical Application – Coronary Stent

The ability of the RS-LEFM procedure to predict stress intensity factors in a practical application is demonstrated by evaluating a postulated flaw within a representative coronary stent under post-deployment conditions. This case study does not involve a typical “forming” process but is representative of an application resulting in residual stresses due to large-strain loading. In this demonstration a stent is being deployed into a hyper-elastic tube that is used to represent an artery on a bench top test fixture. Figure 8 shows the initial and expanded configurations of the stent as well as representative bending stresses that occur at the time of maximum expansion prior to removing the cylindrical expansion tool.

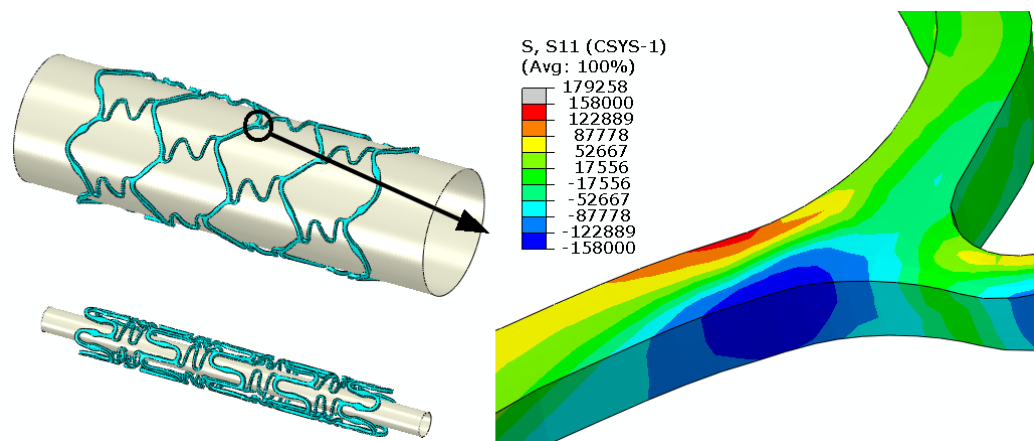


Figure 8: Coronary stent initial and maximum expansion configurations.

The expansion process is a large strain loading step as evident by the significant plastic strains that occur within the stent. Figure 9 shows the plastic strains and residual stresses that are present once the expansion tool has been removed. It can be noticed that the residual stress field is similar to that of a simple bar bent into the plastic range and then allowed to recoil upon removal of the bending load. The surface regions that were in compression during the application of the large strain loads are in residual tension, and those that were in tension are in residual compression. A contribution to the stent stresses is also provided by the radial load exerted on the outside surface of the stent by the mock artery. It is interesting to note that pre-existing flaws which would tend to open within tensile zones during expansion could be evaluated via elastic-plastic J-integral techniques, but post-expansion they would be located in residual compressive zones that tend to keep them closed. However, a pre-existing flaw located in a region of compression during expansion, will remain closed during expansion but will tend to open during the elastic recoil that occurs upon removal of the deployment tool. It is this type of flaw, in a residual tensile zone, for which the RS-LEFM procedure is intended.

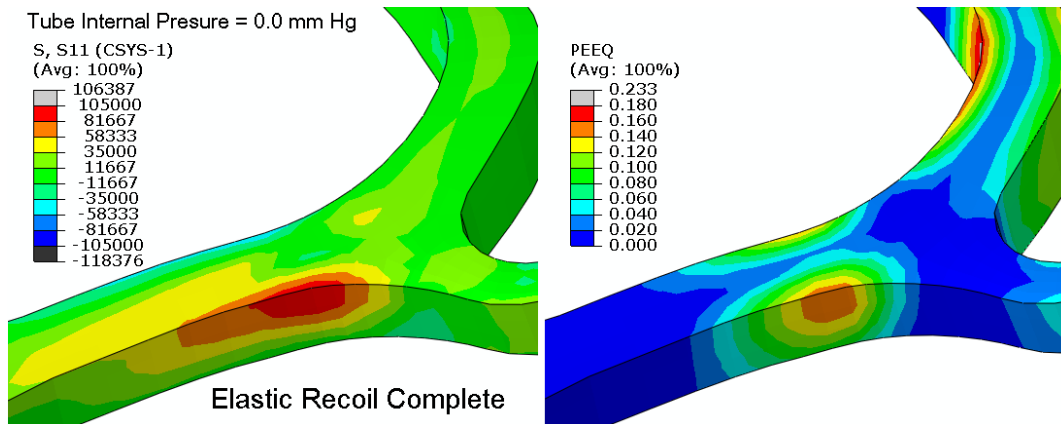


Figure 9: Stress and Equivalent Plastic Strain (PEEQ) after expansion tool removal.

The RS-LEFM procedure was used to evaluate the stress intensity factors associated with a semi-elliptical flaw located within the residual tensile stress zone shown in figure 9. The flaw had a length of 20 microns and a depth of 6.7 microns. The original stent analysis was performed with Abaqus/Explicit using C3D8R elements for the stent and overlaying M3D4R membrane elements on the surface to generate the surface stress and plastic strain results. Since RS-LEFM requires the use of 20-noded bricks to generate quarter point elements, it was not practical to place the flaw into the original stent model and re-run the deployment analysis. A sub-modeling approach was instead utilized to represent the high stress region shown in Figures 9. Figure 10 shows the sub-model region and the basic mesh of C3D20R elements that was used in sub-model testing. The bending stress contour levels shown in Figure 10 are the same as those used in Figure 9.

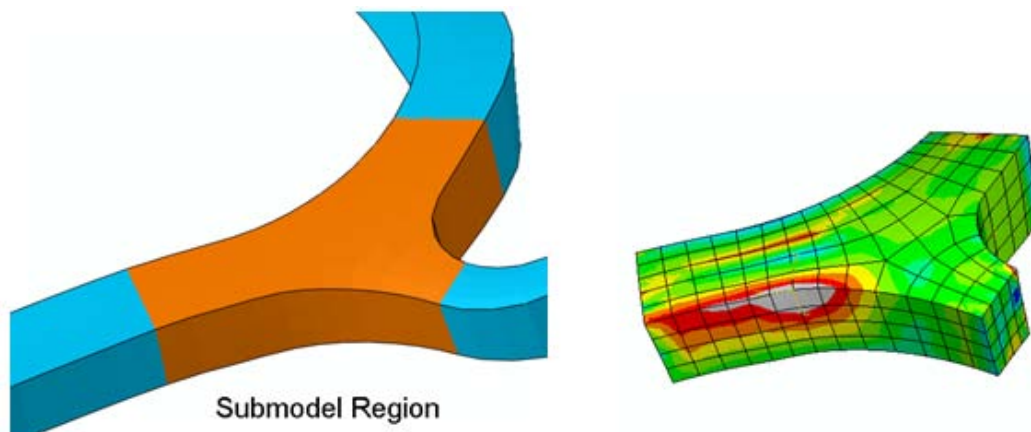


Figure 10: Sub-model region and sub-model test without a flaw.

The stress solution in the sub-model test correlates reasonably well with the original /Explicit analysis, especially when considering that the sub-model utilizes a finer mesh of quadratic interpolated elements as compared to the original model which utilized linear reduced integration elements. The stress “hot spots” along the cut boundaries of the sub-model are a result of the interpolation of boundary restraints from the somewhat coarse global model. These boundary stress artifacts are sufficiently removed from the region where the flaw will be introduced that their influence is negligible. Figure 11 shows the sub-model with the 20 micron side wall semi-elliptical flaw offset towards the outer surface of the stent. The Abaqus surface *TIE capability is used to constrain the fine mesh region containing the flaw to the nominal mesh of the sub-model. The detailed view of the flaw region, with half of the flaw not shown for clarity, has the nodes along the flaw front highlighted. The focused mesh at the flaw front included the quarter point element topology.

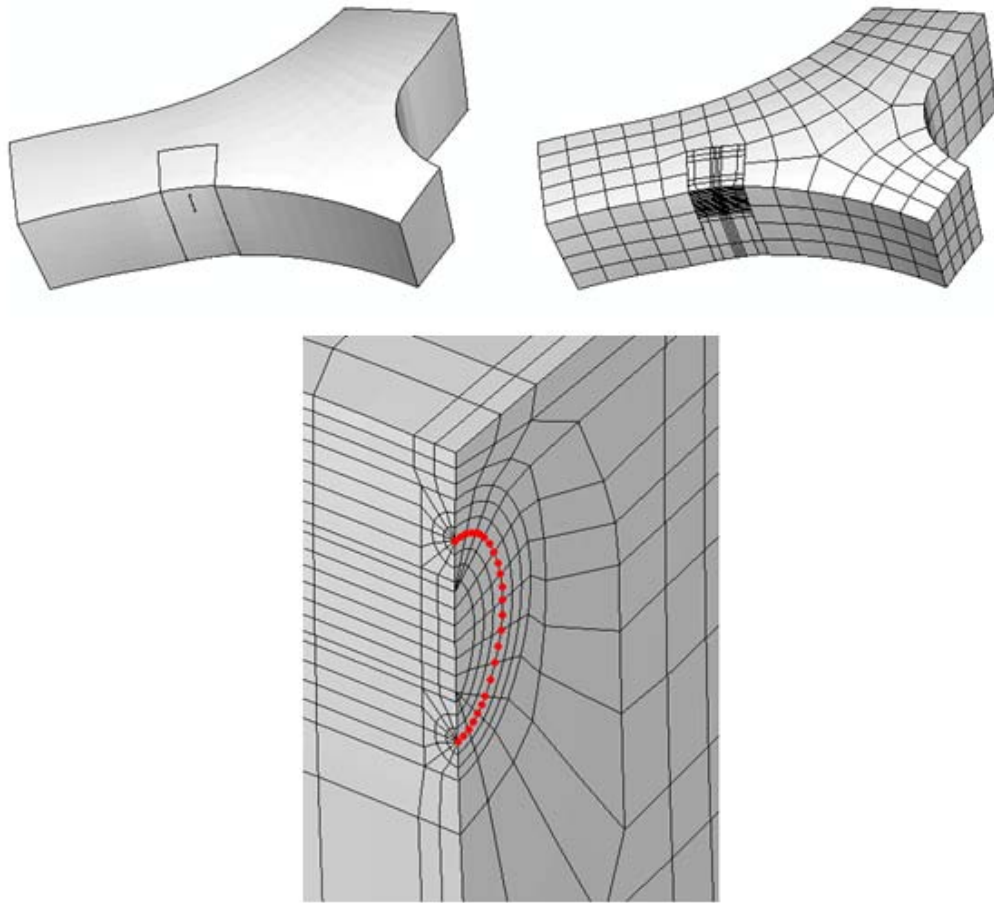


Figure 11: Sub-model region with a 20 micron semi-elliptical flaw.

Figure 12 shows stress contour plots (S11 and Mises) with the flaw held closed and also once the flaw was allowed to open within the residual tensile stress field. The Mises plot at the bottom of Figure 12 contains only the local flaw region mesh with half of the flaw mesh not shown. The singularity associated with the open flaw is clearly illustrated. No plastic zone is present in this model since the material was reset to linear elastic response as part of the final RS-LEFM import analysis.

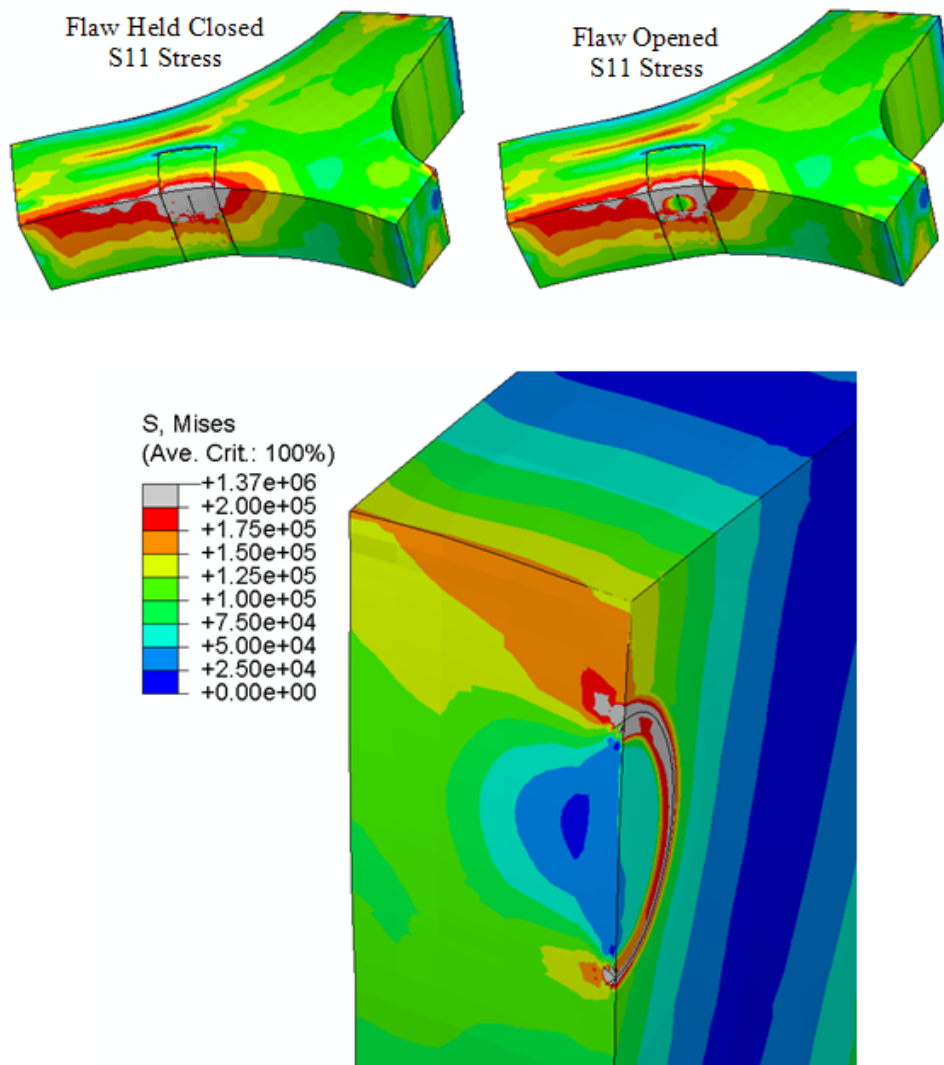


Figure 12: Results for the 20 micron elliptical flaw sub-model import analysis.

The largest Mode I stress intensity factor was $K_I = 2850 \text{ psi}/\sqrt{\text{in}}$, occurring at the upper flaw front where it intersected with the side wall surface. K_I as a function of arc length along the flaw front (top to bottom) is shown in Figure 13. The raw data consisted of values for the alternating corner and mid-side node positions along the flaw arc length. The smoothed curve was generated with the Abaqus/Viewer **smooth(X,I)** curve operation. The integer was set to 1 so that each point on the smoothed curve was generated as the average of the three nearest data points. Figure 14 contains smoothed curves for K_{II} and K_{III} stress intensity factors as a function of arc length along the flaw front.

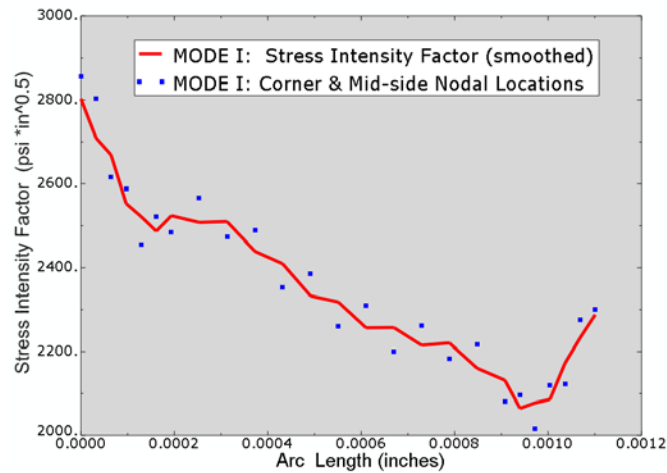


Figure 13: Mode I Stress Intensity Factor vs. Arc Length

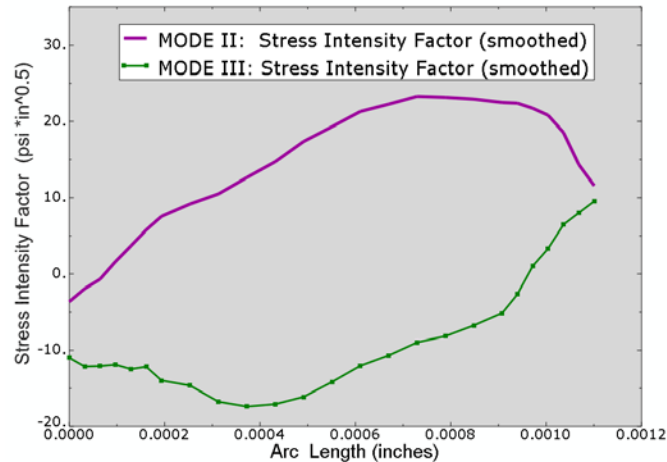


Figure 14: Mode II and III Stress Intensity Factors vs. Arc Length

5. Summary

LEFM stress intensity factors for flaws postulated to exist in regions of tensile residual stress can be calculated by using the Abaqus/Standard import capability. The procedure has been described and validated with respect to a known solution for the Mode I stress intensity factor of a simple bar in 3-point bending. The procedure was demonstrated further by calculating the stress intensity factors (Mode I, II and III) for a 20 micron flaw located in a residual stress zone of a coronary stent in a post-deployed configuration.

6. References

1. Hertzberg, R.W., "Deformation and Fracture Mechanics of Engineering Materials, Fourth Edition," John Wiley and Sons, Inc., New York, 1996, page 757-758.