Fatigue Crack Propagation Comparison of a Hydropower Main Inlet Valve Using 3D Crack Meshes in a Full Model Versus a Sub-model

G. Thorwald, L. Parietti, and D. Blanks

Quest Integrity USA, LLC, 1965 57th Court North, Suite 100, Boulder, Colorado 80301

***Abstract**: Fatigue crack propagation of a surface crack in a hydropower main inlet valve is evaluated and compared by two approaches using Abaqus/Standard. Crack propagation is modeled using a series of 3D crack meshes to compute the crack front stress intensity values, used to compute the crack growth rates. The first approach inserts the crack mesh into the global model and connects it with tied contact, solving the combined mesh including the crack. The second approach uses a sub-model containing just the crack, where the boundary displacements are obtained from the uncracked global model results. The two methods are introduced and compared, first using a cylinder model with a circumferential surface crack. Second, a main inlet valve model with a circumferential surface crack has fatigue crack growth due to cyclic loading from the valve opening and closing. The sub-model's results may differ from the global model as the crack size increases, but it runs more quickly and still provides a useful engineering solution. Model set up effort and analysis run times are compared to help consider when to use each approach. Engineers using Abaqus benefit from having several approaches to model fatigue crack propagation. Computing accurate stress intensity values improves the crack growth rate calculations compared to more conservative crack models. This can help increase the expected life of the component, and could justify fewer inspections during the service life before repair or replacement is needed.*

***Keywords**: Fatigue Crack Growth, Crack Propagation, Cyclic Loading, Hydropower, Main Inlet Valve, Surface Crack, 3D Crack Mesh, Stress Intensity, K_I, J-integral, Tied Contact, Sub-model, Abaqus/Standard.*

1. Introduction

Two common questions about structures with cracks are: will the crack cause it to break immediately, and if not how soon will the crack grow to a size large enough to cause failure? The first question can be addressed using the Failure Assessment Diagram (FAD) method or a ductile tearing instability analysis (Thorwald and Wright 2016, Thorwald and Parietti 2016). Cracks can grow due to fatigue cyclic loading and to address the second question two methods to analyze crack growth propagation will be examined in this paper.

Computing accurate crack front stress intensity K_I values improve the fatigue analysis solution, especially when K solutions are not available for a particular structural geometry or crack location. One way to compute K values in Abaqus/Standard (Abaqus 2016) is by using 3D crack meshes with the crack mesh inserted into the global mesh and connected using tied contact. Another way is to use the 3D crack mesh as a sub-model and impose the boundary displacements from the uncracked global model results. Both of these approaches will be described using two examples: a cylinder and a hydropower main inlet valve (MIV), each with an internal surface crack growing due to cyclic pressure loading.

Figure 1 shows a shop picture of the hydropower MIV before being installed, and the geometry model for Finite Element Analysis (FEA). A half-symmetric model is used for the crack growth analysis. The arrow indicates the direction of water flowing through the valve from the penstock pipe toward the turbine when it is open. The valve operates by rotating a gate (orange and white position indicator in the shop picture) to block or allow water to flow to the turbine; the gate is omitted from the geometry model.

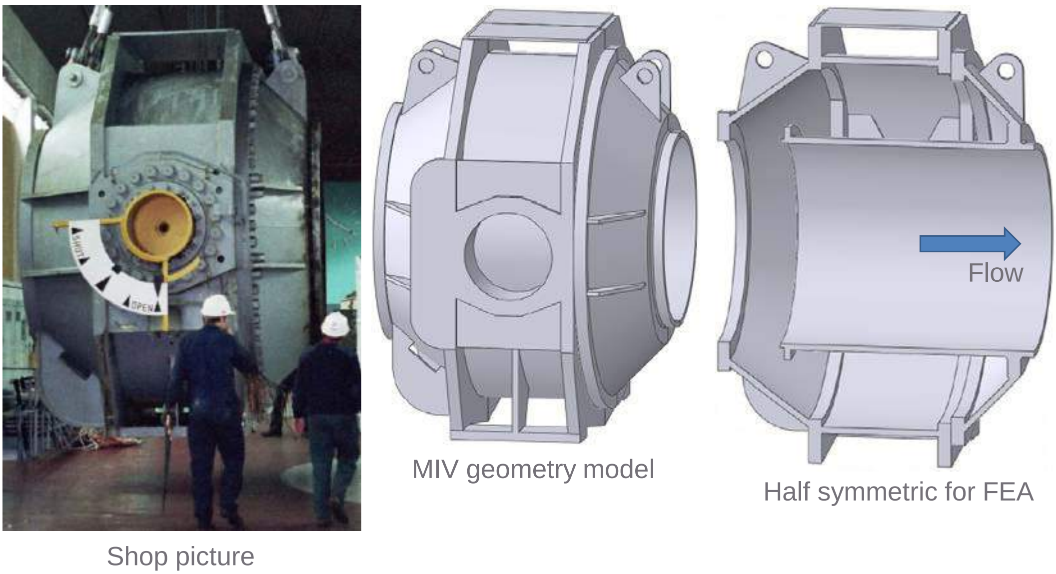

Figure 1. Hydropower main inlet valve, shop picture and geometry model.

1.1 Fatigue Equations

Fatigue crack growth (crack propagation) is described by a da/dN crack growth rate curve, such as the Paris power-law equation:

$$\frac{da}{dN} = C\Delta K^m \tag{1}$$

where *a* is in general a particular crack size, such as the crack length or crack depth, *N* is the number of fatigue cycles, *C* and *m* are material coefficients and ΔK is the crack front stress intensity range (Anderson 2005). For a surface crack the crack length is 2*c* and the crack depth is *a*, shown in Figure 2, and requires solving two simultaneous equations for the crack growth rates.

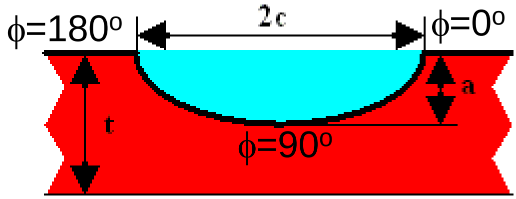

Figure 2. Surface crack profile, crack length 2c, crack depth a.

The crack growth rates are solved incrementally using a re-meshing method (FEACrack 2016) where the crack front ΔK values at the crack tip ($\phi = 0, 180^o$) and crack depth ($\phi = 90^o$) locations are used to update the number of cycles and get the crack size increments for the next crack mesh. Since there are three unknowns: Δa (crack depth increment), Δc (crack length increment), ΔN (increment number of cycles), and only two equations, a small Δa increment size is chosen to provide accuracy in integrating the equations. The crack length incremental growth rate Δc/ΔN is given by:

$$\frac{dc}{dN} = \frac{\Delta c}{\Delta N} = C\left(\Delta \mathrm{K}_{\phi-0,180}\right)^m \quad (2)$$

where $\Delta \mathrm{K}_{\phi-0,180}$ is the crack front stress intensity range at the crack tip (or both crack tips depending on symmetry). The crack depth incremental growth rate Δa/ΔN is given by:

$$\frac{da}{dN} = \frac{\Delta a}{\Delta N} = C\left(\Delta \mathrm{K}_{\phi-90}\right)^m \quad (3)$$

where $\Delta \mathrm{K}_{\phi-90}$ is the crack front stress intensity range at the crack depth location.

The fatigue solution loop in Figure 3 shows the sequence of generating the crack mesh, solving the linear elastic FEA using Abaqus/Standard, and updating the crack growth rates to get the number of fatigue cycles and crack size increment to update the crack size for the next loop increment. Typical fatigue analyses can use several dozen to a few hundred increments. A typical Δa increment size is one percent of the geometry thickness, or smaller for integration accuracy.

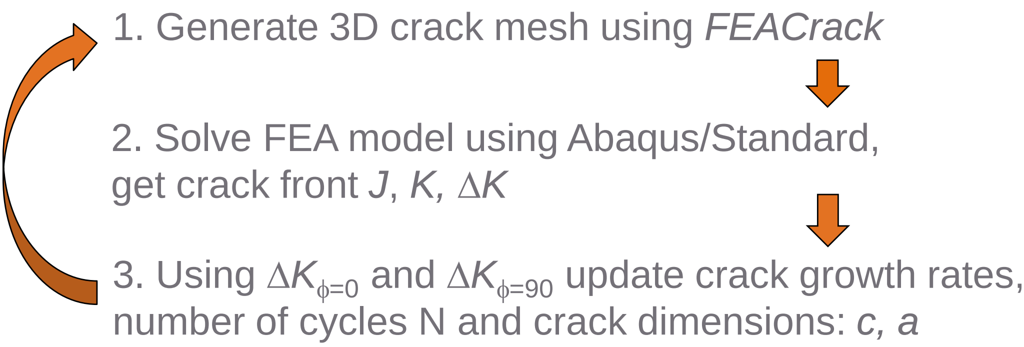

Figure 3. Fatigue crack growth incremental solution loop diagram.

For each increment of the fatigue crack growth re-meshing analysis, either the crack mesh can be inserted into the global model and connected using tied contact (combined mesh), or the crack mesh sub-model has boundary displacements imposed from the uncracked global model results. The trade-off is run-time and accuracy of the K results. Typically the combined crack plus global mesh size is large and takes longer for the FEA solution, but gives the most accurate K results. The crack mesh sub-model is generally much smaller and the FEA solution runs more quickly, but the K results may differ from the combined mesh K results as the crack size increases relative to the size of the sub-model.

2. Cylinder Model

A cylinder with an internal surface crack provides an example for comparison of these two approaches and to a complete cylinder crack mesh not needing tied contact. Figure 4 shows the half-symmetric cylinder global mesh (red) combined with the internal surface crack mesh (blue).

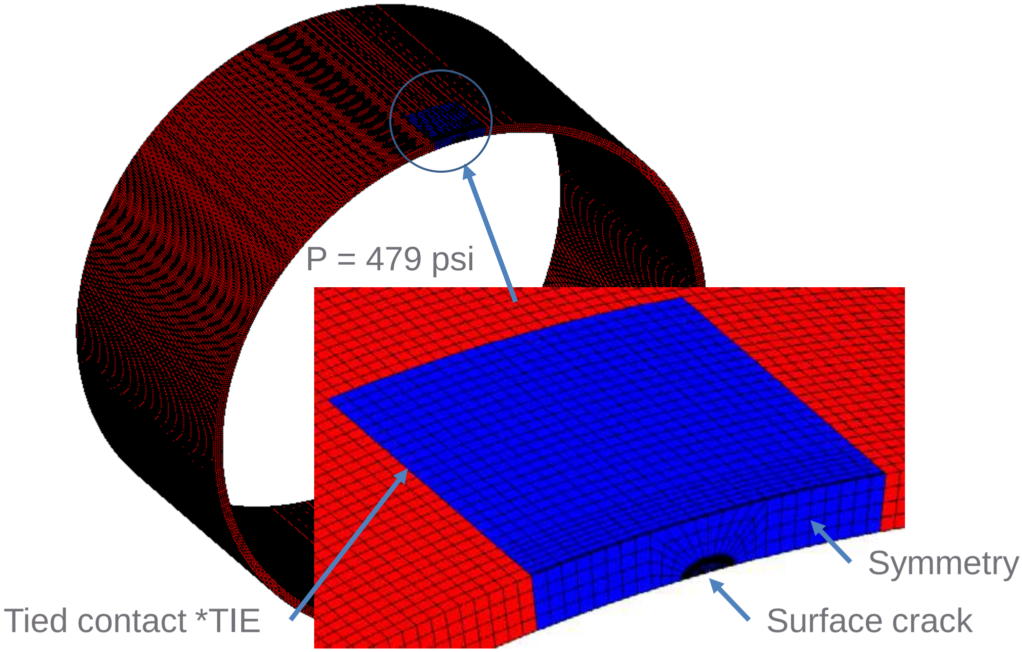

Figure 4. Half-symmetric cylinder combined with and internal surface crack, connected using tied contact.

The two meshes are connected using tied contact around three of the crack mesh edges. Symmetry constraints are applied to the front of the cylinder and crack mesh. The cylinder inside diameter is 130 in, and wall thickness is 2.25 in, which is the same size as the MIV shell size for comparison as an approximation of the MIV geometry. The internal pressure varies from zero to

479 psi for a constant amplitude cyclic load (MIV open or closed); a variable amplitude cyclic load could also be applied if available. The internal pressure is also applied to the crack face. The initial crack length 2c is 1.8 in, and the initial crack depth is 0.45 in. 60 increments were used for this fatigue analysis. The fatigue analysis was also run with a smaller increment to verify the solution didn't change and that 60 increments were adequate. The cylinder global model geometry and mesh are created using Abaqus/CAE.

Figure 5 shows a close up of the surface crack (lower right) and the focused mesh pattern (upper left) used to compute J-integral values in each contour. Since an elastic analysis is used the J-integral values are converted to the equivalent stress intensity K value for use in the fatigue analysis. The upper left picture is from a rotated view inside the cylinder to show the focused mesh pattern needed to compute the J-integral.

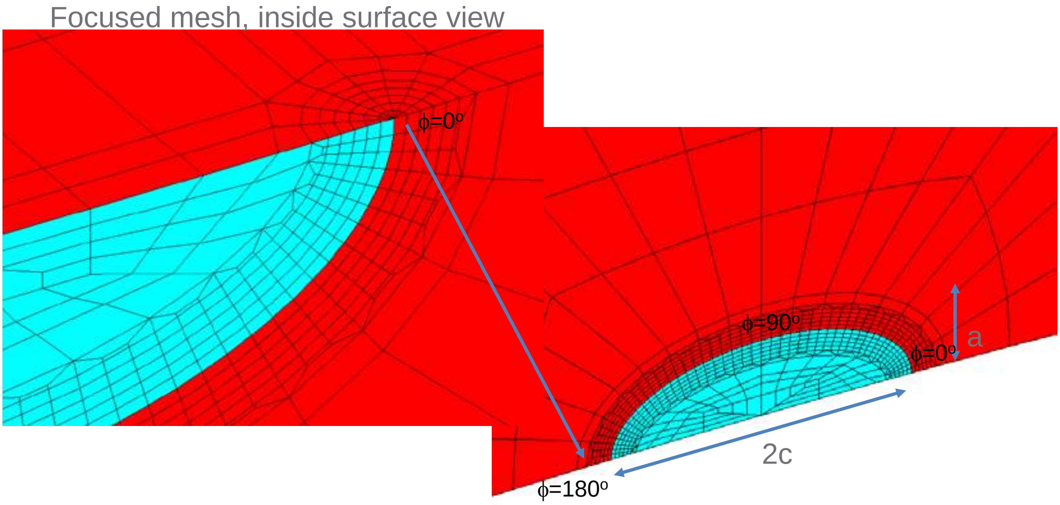

Figure 5. Close up of the surface crack and focused mesh pattern.

The other analysis approach uses the crack-mesh as a sub-model, shown in Figure 6. The crack mesh is the same as in the combined approach, the difference is to apply imposed displacements on the three edges of the crack mesh instead of tied contact. The imposed displacements are obtained from the uncracked global cylinder mesh result file and applied by Abaqus/Standard to the selected crack mesh boundaries during each analysis. The global model does not change during the fatigue analysis. The sub-model size is about five times the thickness for this comparison. Depending on the particular geometry the sub-model size could be larger if desired, and will be compared below.

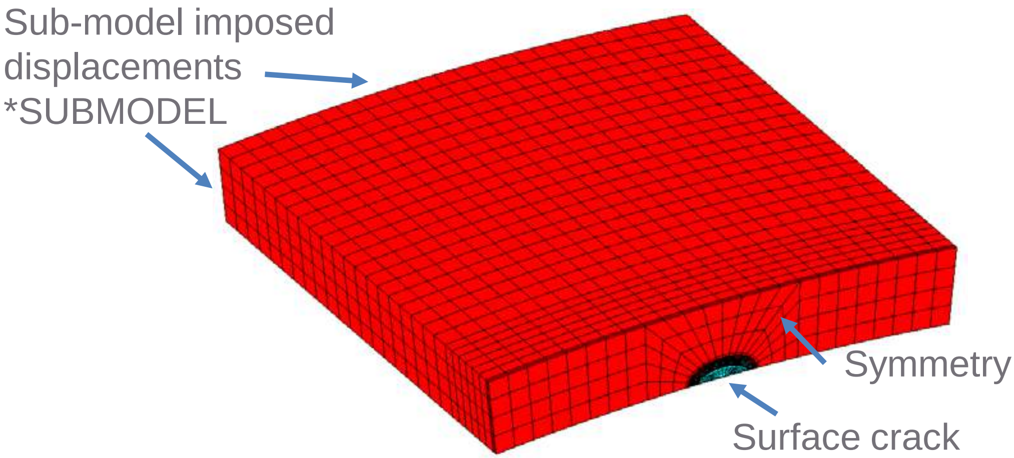

Figure 6. Crack mesh sub-model.

The fatigue analysis runs from the given initial crack depth to a final crack depth of 80% of the thickness in this case. The Paris fatigue equation coefficient *C* is 9.713E-19 and *m* is 3 (ksi-in units). Results from both approaches are shown in Figure 7. The top blue curves compare the crack length results, and the bottom red curves compare the crack depth results. The solid curves are for the complete cylinder crack mesh built directly by FEACrack that does not need tied contact which allows another comparison, the dotted curves are for the crack mesh combined with the global mesh, and the dashed curves are for the crack mesh sub-model. The results show close agreement in the crack sizes until later in the analysis when the crack size has increased. There is a small difference between the complete cylinder crack mesh and the combined crack mesh, perhaps due to small differences in the mesh node coordinates or the use of tied contact in the combined model.

We have observed that the crack sub-model K values tend to underestimate the crack sizes later in the fatigue results compared to the combined mesh. As the crack size increases relative to the sub-model it reduces the sub-model's stiffness and since the imposed boundary displacements remain the same, this tends to reduce the sub-model crack front K values. For an engineering assessment the decrease in accuracy of the fatigue solution using a sub-model may be offset by the benefit of a faster run time to get a solution. For example, this analysis shows that around a million cycles are needed for the crack depth to approach 80% of the thickness, and knowing that fatigue duration more quickly using the sub-model approach may allow for an operational decision to continue in-service without an unscheduled shut down.

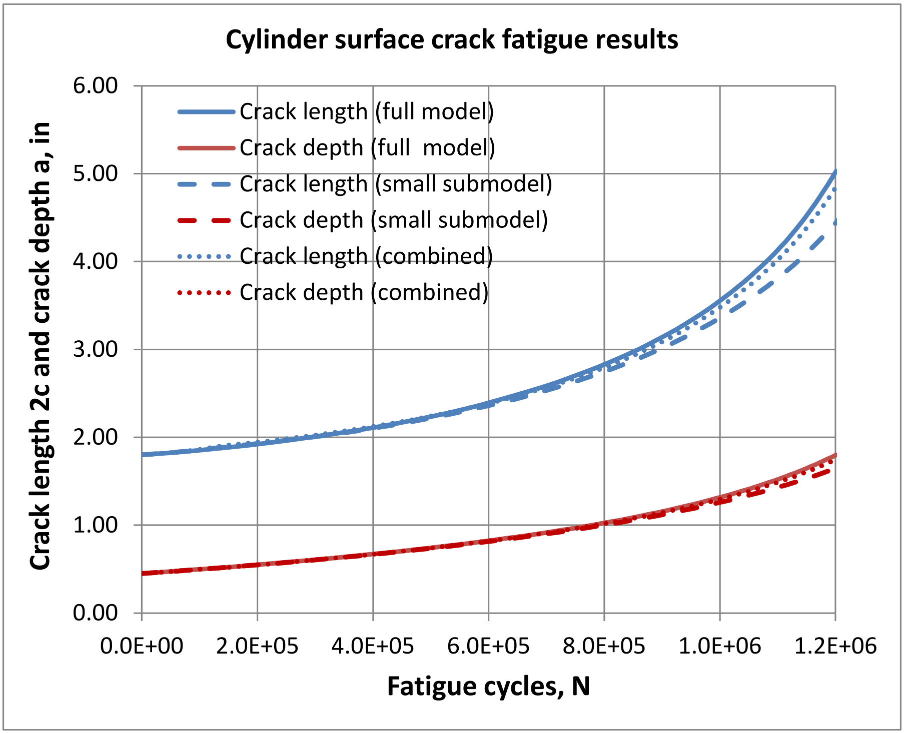

Figure 7. Cylinder surface crack fatigue results.

Figure 8 shows the total run time comparison for the small sub-model versus the combined mesh (the large sub-model is described below). The combined mesh takes a little over five times as long to run as the sub-model.

The combined mesh has 485438 nodes, the small sub-model has 35439 nodes, and the large sub-model, see Figure 9, has 80079 nodes. All the crack meshes use 20-node brick elements with reduced integration.

The model set up effort is a little more for the inserted crack mesh combined with the global model since the region where the crack is inserted needs to be partitioned and removed from the global model. The global model input file also needs to have the tied contact surfaces selected to be connected to the crack mesh. Generating the crack mesh is the same effort for either analysis approach.

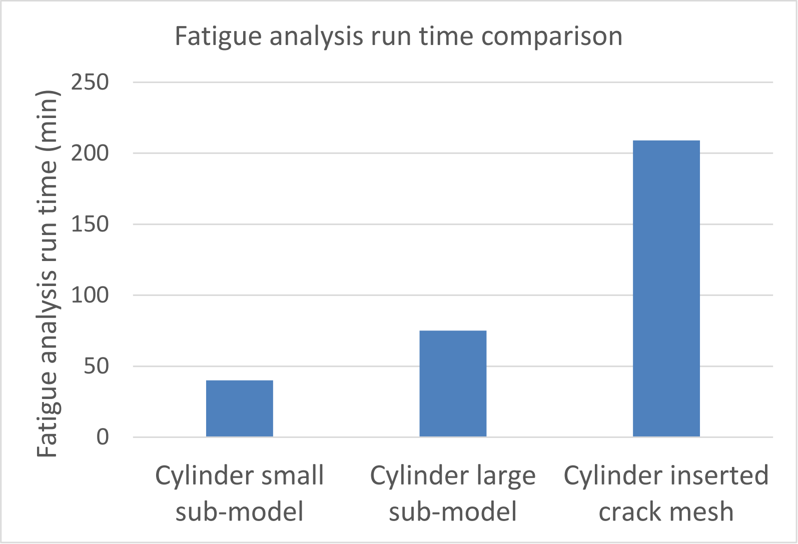

Figure 8. Cylinder fatigue analysis total run time comparison.

To examine the effect of the sub-model size on the fatigue results, a larger sub-model with a size about 10 times the thickness compared to the size of the small sub-model has been analyzed, and is shown in Figure 9. The thickness and initial crack size are the same as the small sub-model.

The dotted curves for the large sub-model fatigue results in Figure 10 shows closer agreement to the combined crack plus global cylinder mesh. This improvement in fatigue results appears to support the idea that if the crack size is relatively small compared to the sub-model size, then the sub-model's stiffness is not affected as much giving more accurate K results.

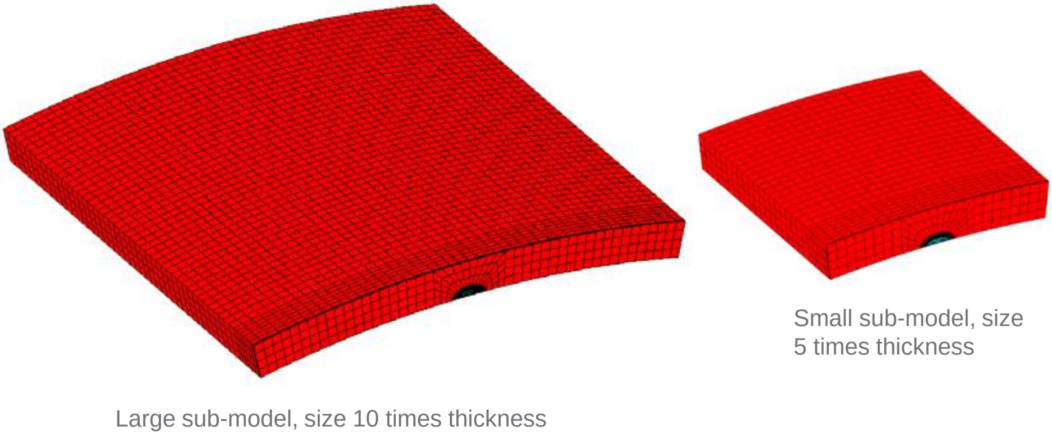

Figure 9. Larger sub-model size for fatigue results comparison.

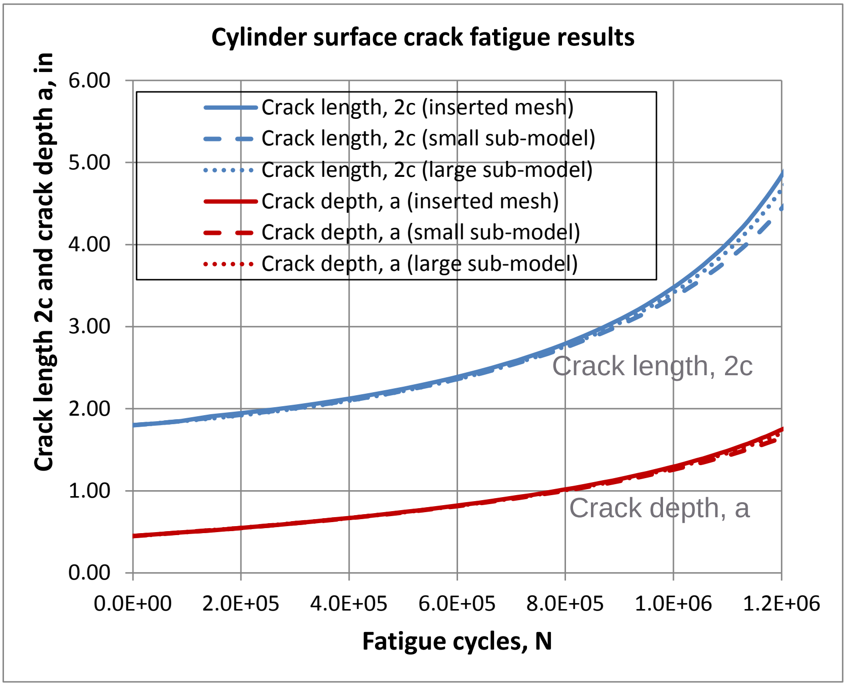

Figure 10. Fatigue results comparison with large sub-model.

3. Main Inlet Valve Model

The hydropower MIV, shown in Figure 1, with an internal surface crack is analyzed using the same two methods of an inserted crack mesh and a crack mesh sub-model. The principal stress results of the uncracked MIV model are shown in Figure 11. The MIV geometry and mesh were created using Abaqus/CAE. The crack is located at the toe of the inner fillet where the valve shell connects to the frame and is one of the locations in the MIV with locally higher stress.

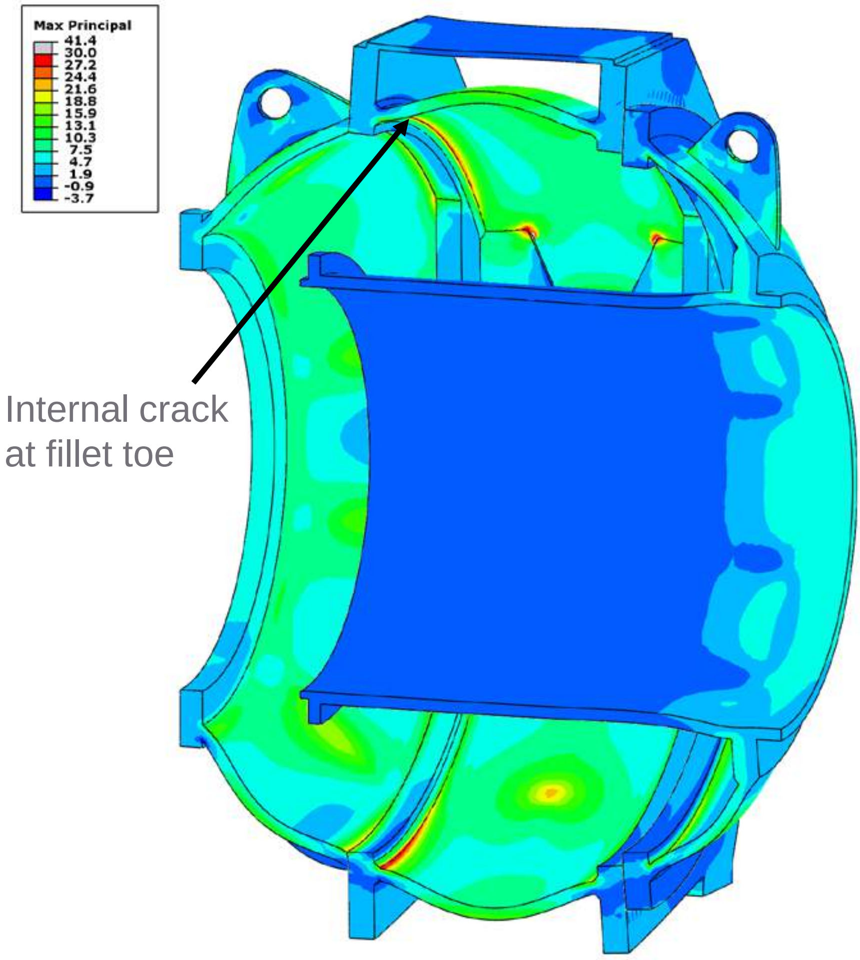

Figure 11. Internal surface crack location in hydropower main inlet valve, principal stress results for the uncracked model.

The half symmetric MIV with inserted crack mesh for fatigue analysis is shown in Figure 12. The surface crack is located at the toe of the inner fillet where the valve shell connects to the outer frame and is oriented circumferentially. The symmetry plane is across the middle of the surface crack so half the crack length is included in the mesh with both crack faces.

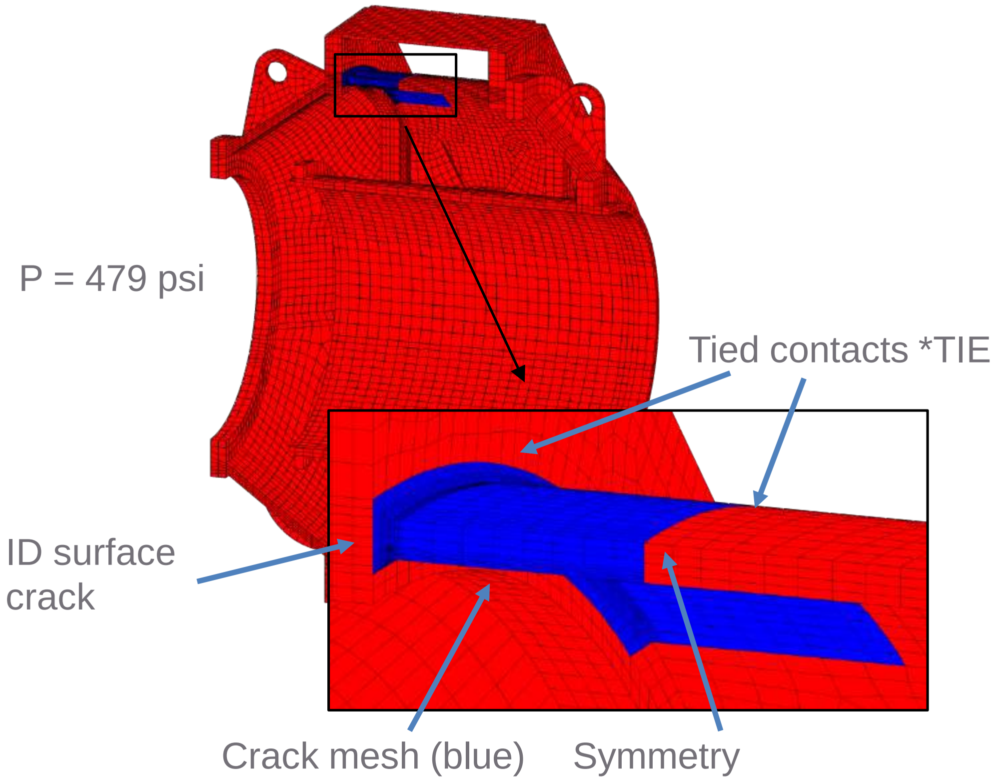

Figure 12. Hydropower main inlet valve with inserted crack mesh for fatigue analysis.

The crack mesh sub-model is shown in Figure 13 with a close up of the crack focused mesh (in blue). The MIV shell inside diameter is 130 in, and wall thickness is 2.25 in (same as the cylinder models), and the internal pressure varies from zero to 479 psi for a constant amplitude cyclic load when the valve gate opens and closes. The same initial surface crack size and same Paris fatigue coefficients as the cylinder are used to allow results comparison.

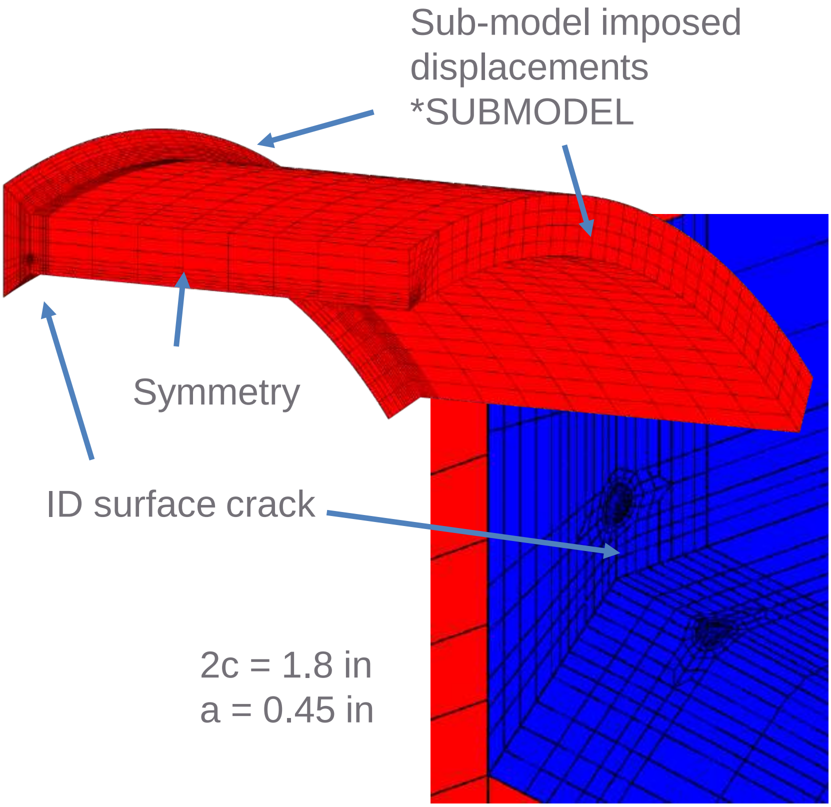

Figure 13. Crack mesh sub-model for the main inlet valve, close up of focused mesh in blue.

The MIV fatigue results in Figure 14 show a larger difference between the combined mesh and the crack sub-models. One factor may be that the crack is located near the side of the sub-model at the toe of the inner fillet, which may cause a greater effect from the sub-model imposed boundary displacements. Since the crack tip is at the base of the inner fillet, it is within the locally higher stress region, which contributes to the faster crack length growth rate. The dotted and dashed lines show the sub-model fatigue results, and the larger sub-model shows the trend to improve the fatigue results compared to a smaller sub-model.

The MIV fatigue analysis run time values are shown in Figure 15, and in this case the combined mesh takes 2.4 times as long to run as the small crack sub-model. The combined MIV mesh has 245523 nodes, the small MIV sub-model crack mesh has 42473 nodes and the larger size sub-model also has 42473 nodes.

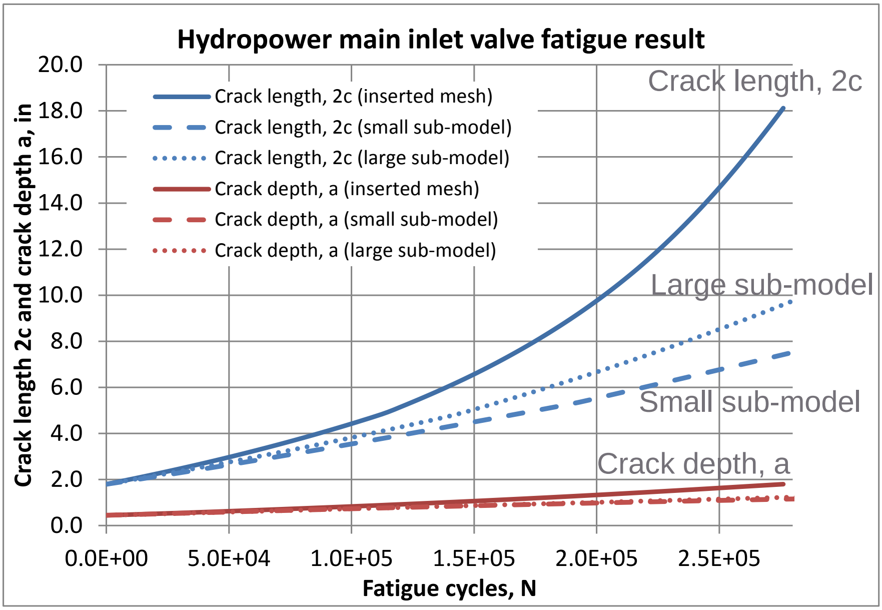

Figure 14. Hydropower main inlet valve internal surface crack fatigue results.

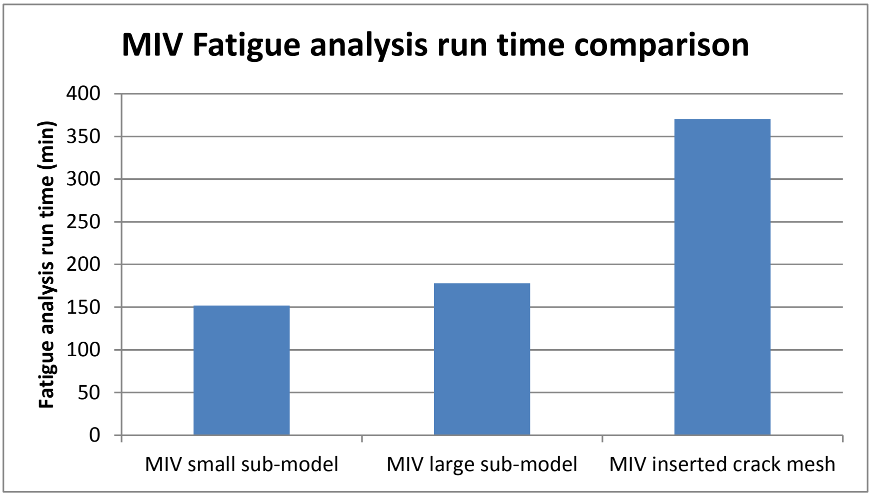

Figure 15. Main inlet valve fatigue run time comparison.

The hydropower MIV and cylinder fatigue results are compared in Figure 16 using the inserted crack mesh method result values. The internal surface crack grows more quickly in the MIV than in the cylinder. Since the surface crack is located at the toe of the inner MIV fillet it is in a locally higher stress region, which increases the crack length growth rate compared to the cylinder that has just hoop stress. In this case the cylinder model with the same size as the valve shell does not provide an adequate estimate of the fatigue crack growth, so inserting the crack mesh into the MIV is used to get a more accurate fatigue analysis. The MIV sub-model would provide a better estimate of the crack growth than the cylinder if time to obtain a result was limited. Both approaches can be used in an assessment project by using the sub-model to obtain initial fatigue results quickly, and then running the combined model to confirm the fatigue results, perhaps by running a longer fatigue analysis overnight. The next task in this assessment would be to check the final crack size for failure, such as using the FAD.

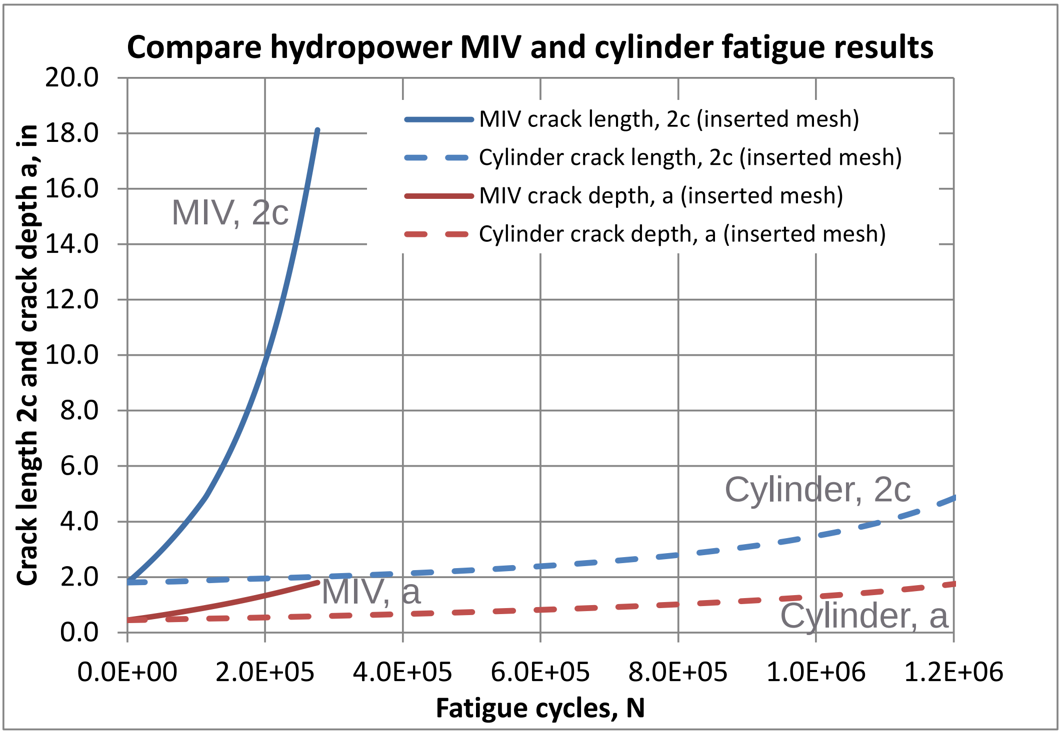

Figure 16. Compare hydropower main inlet valve to cylinder fatigue results.

4. Summary

Fatigue crack growth propagation analysis can be done using two approaches using Abaqus/Standard: the crack mesh inserted and combined with the global mesh, or the crack mesh as a sub-model. The sub-model fatigue results tend to underestimate the crack growth as the crack size increases relative to the sub-model size. The crack mesh sub-model runs faster than the combined mesh so results can be obtained sooner, which may help support quicker operational decisions to continue to operate or shut down for repair. If possible, a larger sub-model size will help reduce the difference in fatigue results compared to the combined mesh. Depending on the structural geometry a simplified case may not provide an adequate estimate of the fatigue crack growth. In this case the hydropower main inlet valve has a fillet that increases the local stress and causes faster crack growth compared to the cylinder model. Inserting the crack mesh into the main inlet valve improved the fatigue results. The inserted crack mesh method gives more accurate results, but will take longer to run, depending on the total mesh size. Having two fatigue analysis approaches benefits engineers by allowing the choice of a method to suit each analysis project.

5. References

1. Abaqus User's Manual, Abaqus 3DEXPERIENCE 2016, Dassault Systémes Simulia Corp., Johnston, RI.
2. Anderson, T. L., Fracture Mechanics: Fundamentals and Applications, 3rd ed., 2005, CRC Press, Taylor & Francis Group, section 10.2, equation 10.5.
3. FEACrack, version 3.2.30, 2016, Quest Integrity USA LLC, 1965 57th Court North, Suite 100, Boulder, Colorado, 80301, www.questintegrity.com.
4. Thorwald, G. and J. Wright, "Ductile Tearing Instability Assessment of a Cracked Reactor Pressure Vessel Nozzle for Larger Critical Crack Size Compared to the FAD Method,", 2016 SIMULIA Science in the Age of Experience conference, May 2016, Boston, MA
5. Thorwald, G. and L. Parietti, "Leak-Before-Break Assessment of a Cracked Reactor Vessel Using 3D Crack meshes," Proceedings of the ASME 2016 Pressure Vessels and Piping Conference, PVP2016, July 17-21, 2016, paper no. PVP2016-63910.