Using Co-Simulation to Extend the Usage of XFEM

Lyonel Reinhardt, J.A. Cordes, David Geissler

U. S. Army

Abstract: XFEM, Extended Finite Element Method, can accurately predict fracture failure. The Army would like to use XFEM in many areas to prevent fracture failures as part of the design process. Problems of interest include drop tests, concrete penetration, gun launch, recoil, and interaction of gun supports with ground. These problems have loads that are dynamic and multi-directional. The geometry often requires millions of degrees of freedom for accuracy. Explicit codes are generally used to capture the complicated system response as a function of time.

Currently, Abaqus Explicit does not support the use of X-FEM. This can be overcome with co-simulation. Co-simulation is the use of two simultaneous analyses to solve one problem. For this study, co-simulation combined dynamic explicit analysis with dynamic implicit analysis and X-FEM. Fracture was simulated on a grenade assembly during a drop test. Co-simulation results were similar to an implicit dynamic analysis with XFEM. The co-simulation was completed in about 3% of the time required for the implicit dynamic analysis alone.

Keywords: co-simulation, implicit solver, explicit solver, fatigue, failure, fracture, XFEM

INTRODUCTION

Fracture and failure are common problems with Army equipment. The ability to predict fracture improves the reliability of equipment, increases safety, and saves money. Much of the analysis done on Army equipment is dynamic in nature. Unfortunately, Abaqus does not support XFEM in its explicit dynamic code. By using co-simulation, XFEM can be extended to the Abaqus explicit code.

An M67 hand grenade drop test was used to evaluate the co-simulation technique. Co-simulation with XFEM was compared to a dynamic implicit analysis with XFEM. First, simulations were completed with Abaqus Implicit and

XFEM. Second, modifications to the critical part were made which divided the part into two tied regions. The dynamic implicit analysis was repeated to assess the effect of tied regions near an enhanced XFEM region. Finally, the model was copied, two separate models were made, and a co-simulation was run. The XFEM region remained a dynamic implicit analysis. The rest of the grenade assembly was evaluated using explicit analysis. The border between the two regions was defined as part of the co-simulation. The resulting stresses and strains were similar. The location and extent of crack growth was similar. The co-simulation model ran in about 3% of the time compared to the dynamic implicit analysis alone. This paper describes the models, the method, and provides tips for conducting a co-simulation with XFEM and Abaqus Explicit analysis.

BACKGROUND

In previous analysis [1] the Army validated the accuracy of XFEM for thin aluminum panels with angled cracks. In that study, the panels were modeled using Abaqus Standard with XFEM. The direction of crack growth was accurately predicted. The failure loads were predicted within 5% of experimental values.

MODEL

General Purpose Finite Element Analysis

For this study, ABAQUS Standard 6.10.EF1, 6.11PR1 [2] was used for the dynamic implicit model. ABAQUS Explicit 6.10.EF1, 6.11PR1 was used for the explicit calculations in the co-simulation.

Statement of Problem

The M67 hand grenade (Fig. 1) is dropped from a height of 1.219 meters (4 feet). In actual tests, the grenade is dropped in several orientations to assure safety and functionality.

Figure 1 - M67 hand grenade

Initially, the explicit finite element analysis was used to determine the worst-cases orientation for drop test. The model included 9 parts. It was determined that the highest stress in the handle occurred with the grenade oriented to hit the ground on its corner, Figure 2. The stress concentration in the notched region of the handle (near the pin in Figure 2) was one of the regions of concern for possible fracture failure. This case was used for the study in the present paper.

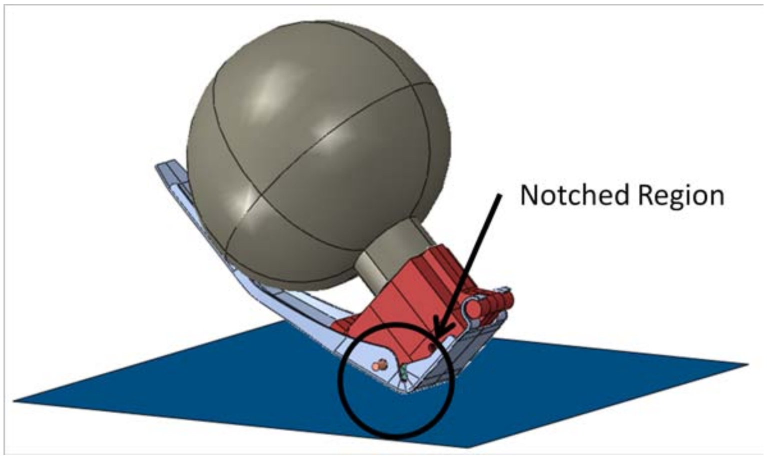

Figure 2-Drop Test Model

Material Models

The material properties for a dynamic analysis of the M67 handle are given in Table 1. The material models for other parts of the grenade are omitted from this discussion. Except for the handle, all materials were modeled as linear elastic.

Table 1 Material Properties, M67 Handle

Yield Strength, MPa	*305.3*
Ultimate Tensile Strength, MPa	437.7
Young's Modulus, GPa	204.8
Strain at Ultimate Failure, percent	18.0
Poisson's ratio	0.29
Density kg/m^3	7,823.0

The handle was modeled with an elastic/plastic material model using the data in Tables 1 and 2. Part of the handle was modeled as an enriched area with cohesive elements. A damage criteria and damage evolution criteria were specified with the handle material model. For this study, the maximum principal stress criteria was chosen for the damage criteria ('Maxps Damage' in Abaqus). The ultimate tensile strength was used for the maximum principal stress. Damage evolution was based on an energy criteria equivalent to the strain energy release rate, G_I. The damage energy was estimated $G_1 = K_{IC}^2/E$, or 5.3. kN/m.

Table 2 Material Failure Model

ASTM A109 Steel	
Maxps Damage	Max Principal Stress: 345 MPa
Damage Evolution	Type: Energy
	Softening: Linear
	Degradation: Maximum
	Mixed mode behavior: Mode-Independent
	Mode mix ratio: Energy
	Fracture Energy: 5.3-kN/m

Boundary Conditions, Loads, and Constraints

In the three models described below, a deformable grenade assembly was dropped onto a rigid surface. The rigid surface was constrained in all directions (Fig. 3) and the grenade assembly was given an initial velocity to simulate the drop height. Modeling the grenade drop from its initial height and zero velocity would have been computationally expensive. Calculating the velocity of the grenade just before impact allowed the grenade to be positioned just above the impact surface, saving considerable computation time.

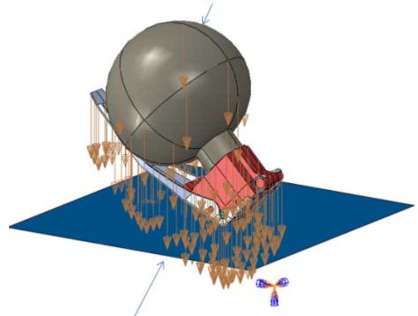

Figure 3-Initial conditions assigned to assembly

Specify XFEM in ABAQUS

Three models were used to validate the co-simulation method. After validation XFEM was added to the implicit dynamic tied model and the implicit dynamic co-simulation model. The following tips are provided for using XFEM with implicit dynamic analysis.

1. In the implicit model, create an XFEM enriched area in the part of interest, that is the part or area where cracking is expected. The entire part can also be chosen.
2. To define an enriched area go to the interaction_module/special/crack/manager.
3. In the crack manager window, click create.
4. In the create crack window, select XFEM.
5. Select the instance, cell or elements for the enriched area. Note for co-simulation don't select elements on the co-simulation interaction face (figure 10), the interaction load will initiate cracks there.
6. Add damage to the material model (table 1) used in the enriched area (table 2).
7. Add the following field output variables, STATUSXFEM, PHILSM
8. In the Step module select ‘moderate dissipation’ for a dynamic implicit analysis

Implicit Model & Co-Simulation Model

Results from two analyses were compared. The grenade handle was cut into two separate parts to later facilitate co-simulation. The implicit model was changed to have a smaller enriched section near the stress concentration in the handle, Figure 4. The parts were then tied together. Results were run with Abaqus Implicit and XFEM.

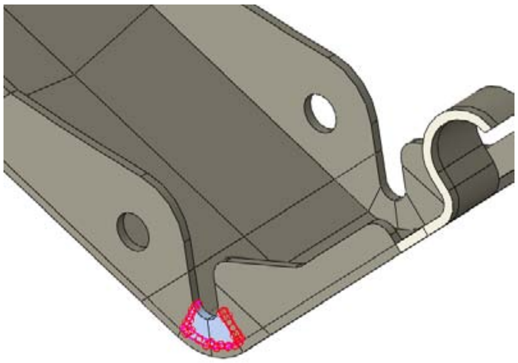

Figure 4-Grenade handle separated into two pieces

The third model was a co-simulation model where most of the model was run as an explicit model with a single part run as an implicit model. The co-simulated drop test was modeled mostly in explicit. The notched XFEM area of interest was modeled in implicit to implement the XFEM technique, circled in Fig. 5. The material model (table 1) was given a damage model (table 2).

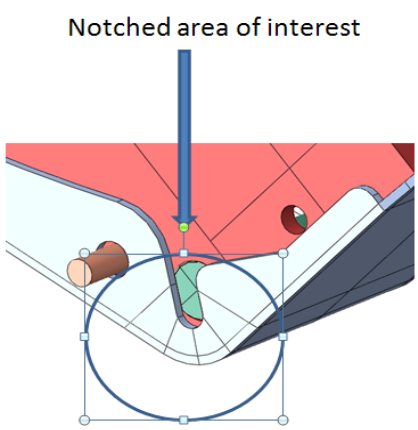

Figure 5-Notched area of interest

Co-Simulation Method (co-simulation)

The following check list is provided for a co-simulation analysis.

1. To setup a co-simulation analysis, create a model where the portion of the part to be co-simulated has been split into two parts and tied together.
2. Make two copies of the model. One model is used for the explicit analysis and one model is for the implicit analysis. The original model is not used, but should be maintained for backup.
3. Convert one of the models, explicit to implicit or implicit to explicit so there is one model of each type.
4. Go to one of the models and in the model tree, right click one of the part instance's for solution in the other model and select Link Instances. For example, if you are in the implicit model select an instance in the explicit analysis.
5. In the link instances window (figure 6) select the model to link to under the model column and be sure to check "Also exclude child instances from the simulation". Then click the link button, this part will now be solved in the other model.

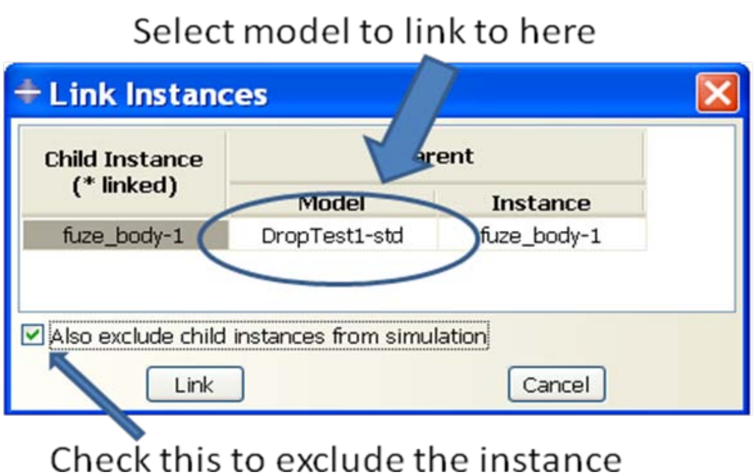

Figure 6 ABAQUS Link Window

6. Repeat this for every part to be linked. Then review all the loads, initial conditions, constraints etc. that relate to the linked parts and remove or modify them, as these parts are no longer in the analysis. For this study, there were no initial conditions, tied constraints, or contacts in the <u>dynamic implicit</u> analysis. All contact and initial conditions are part of the dynamic explicit model.

7. Repeat this for the other model. When complete, all the parts in the explicit analysis that need to be solved in the standard analysis should be linked to the standard analysis. In the standard analysis, the parts to be solved in the explicit analysis should be linked to the explicit analysis. Again, make sure to remove or modify all the loads, constraints, initial conditions etc. that relate to the linked parts.

8. To create the co-simulation interaction, Fig. 7, select one of the models and go to the interaction manager. Create a standard-explicit co-simulation interaction and pick the faces where the interaction will occur, the tied faces shown in figure 4. Click ok and repeat on the other model. This will create the faces that will interact between the two models.

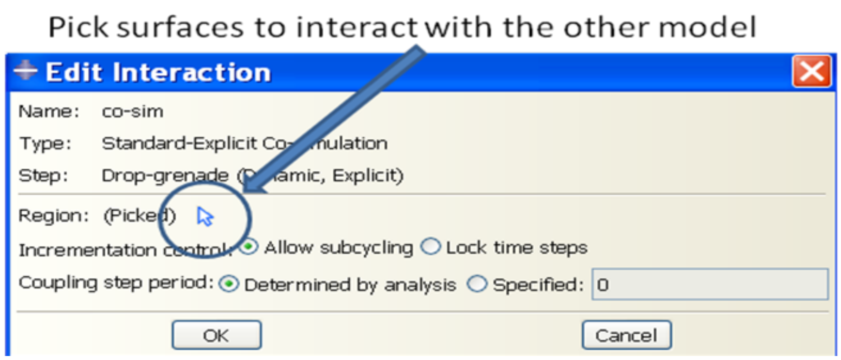

Figure 7. Specify the Surface Interaction between the 2 models in Co-Sim

9. To execute the co-simulation go to the Job module and select the co-execution manager which is two buttons below the job manager. In the co-execution manager window click the create button. In the edit co-execution window, figure 8, under the Model Name select the two models to co-simulate and click the ok button. To start the co-simulation click the submit button on the co-execution manager window.

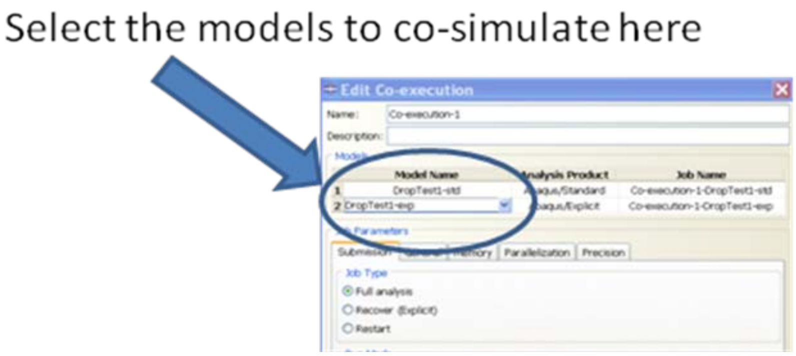

Figure 8 ABAQUS Edit co-execution window

Mesh Details

The mesh had to be fine enough to allow four elements through the thickness of the handle, Fig. 9. The mesh also had to line up between the implicit and explicit regions. The majority of the handle part represented by the blue area is solved explicitly. The small grey area is solved by the XFEM implicit solver, Fig. 10. The XFEM enriched zone was reduced to eliminate elements near the co-simulation boundary, Fig. 11.

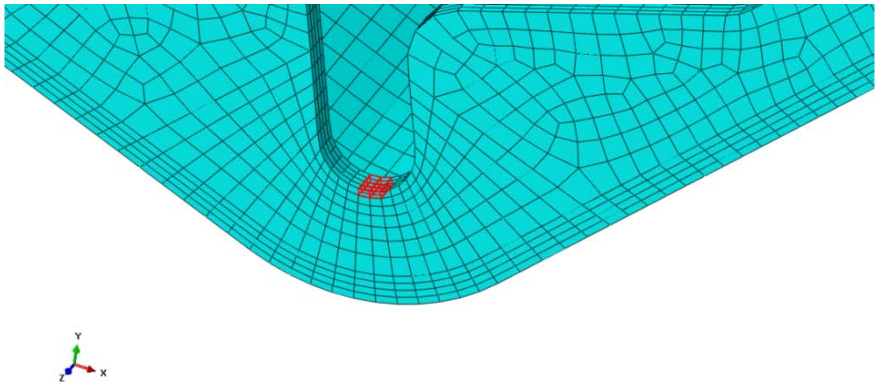

Figure 9-Mesh detail

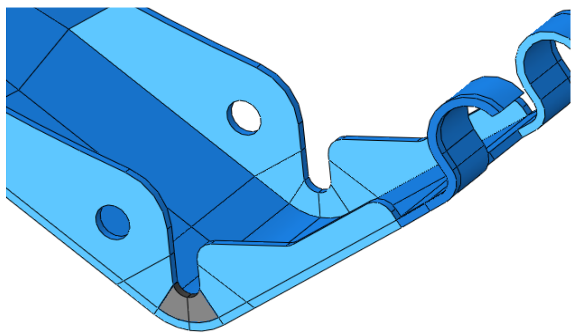

Figure 10-Co-simulation regions

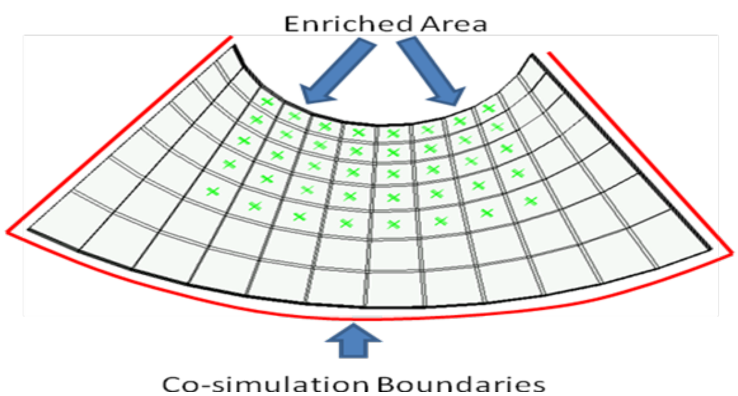

Figure 11-XFEM Enriched Zone

SIMULATION RESULTS

Validation

To validate the method, three cases were compared. Case 1 was a typical mesh on one continuous part (Fig. 12). Case 2 consisted of two separate parts tied together (Fig. 3). Case 3 consisted of two parts co-simulated (Fig. 13). In order to try to eliminate an unrealistic representation of the stresses at the area of interest, eight elements at the same location (Fig. 14) were compared. The results are shown below (Fig. 15). Figure 15a shows equivalent plastic strain over time. Figure 15b shows fairly even tracking of the von Mises stress near the notch. The stresses in the co-simulation, Case 3, are closer to the tied result than the continuous mesh result, Case 1.

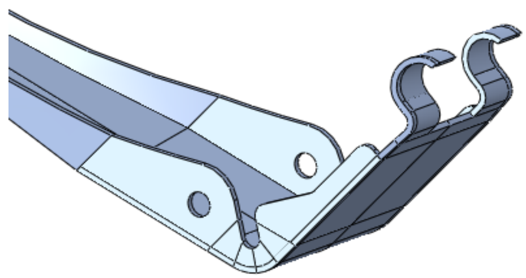

Figure 12-One continuous part, Case 1

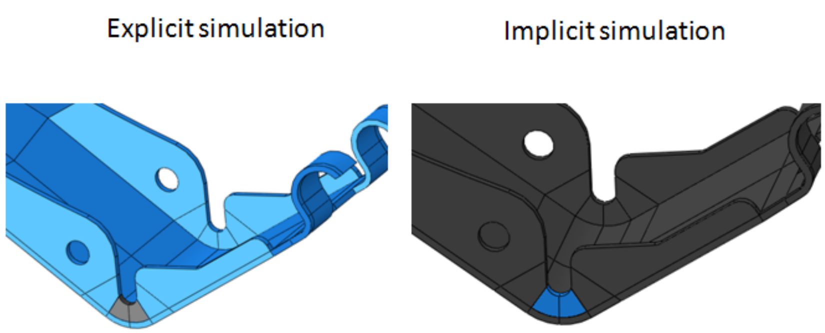

Figure 13-Co-simulation domains, Case 3

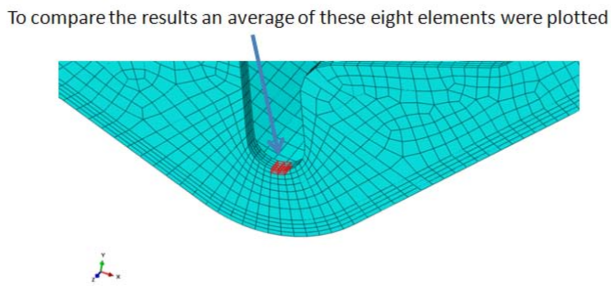

Figure 14-Elements used to determine average stress

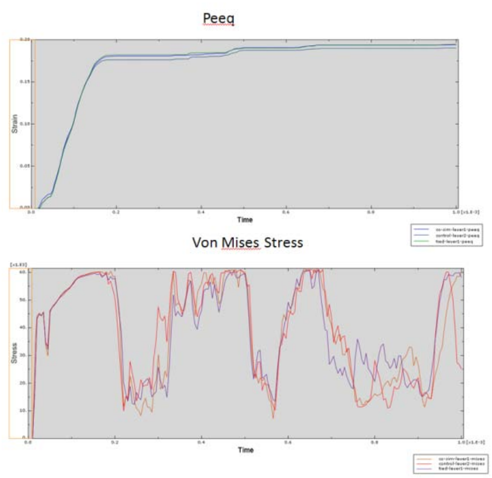

Figure 15-Comparison between continuous mesh, tied, and co-simulated analysis (Cases 1,2, and 3)

XFEM Results

Figure 16 shows a comparison of crack growth between the dynamic implicit analysis and the co-simulation. Total crack growth is similar. Figure 17 shows an interior view of the total crack growth, also similar between the simulations. Figure 18 shows early crack development for the dynamic implicit and co-simulation models. The crack initiation was slightly different. Figure 19-21 shows a comparison of displacements, also slightly different.

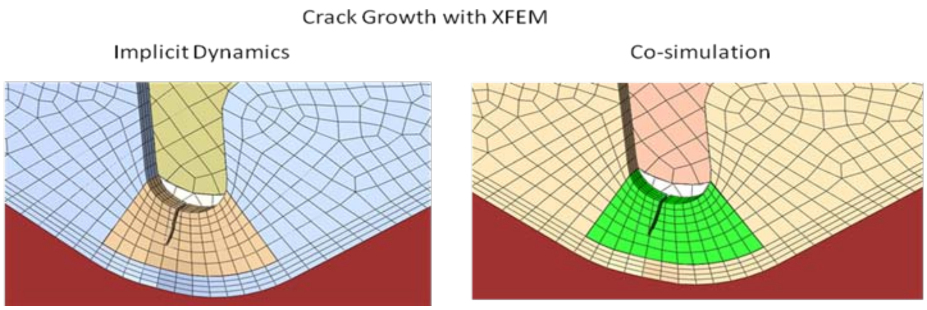

Figure 16-Comparison in crack between implicit and co-simulation

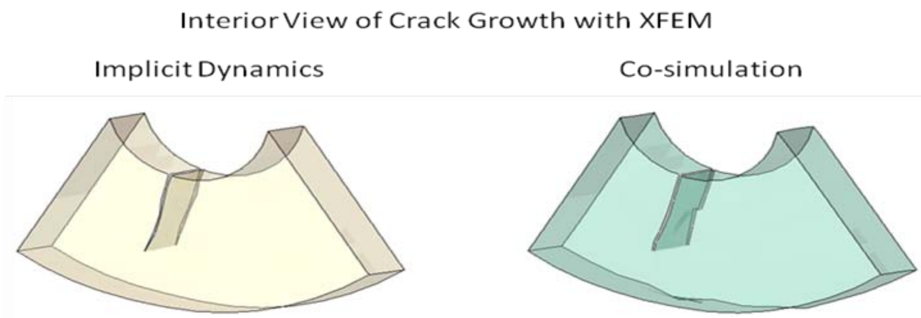

Figure 17-Comparison of interior crack shape between implicit and co-simulation

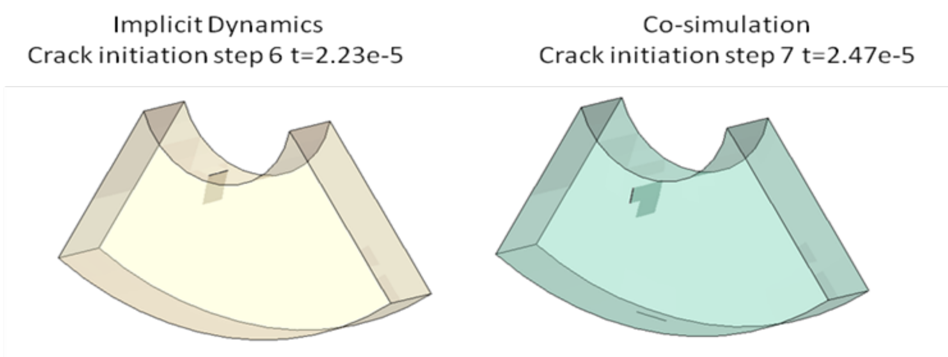

Figure 18-Comparison of crack initiation between implicit and co-simulation

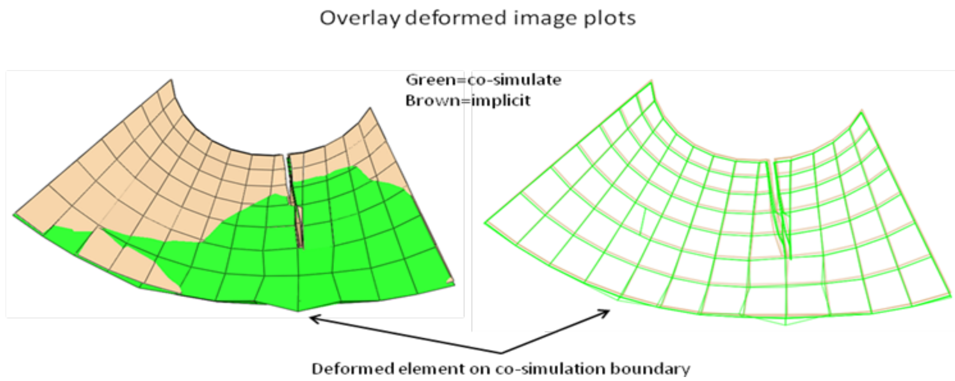

Figure 19-Comparison of Displacement between implicit and co-simulated

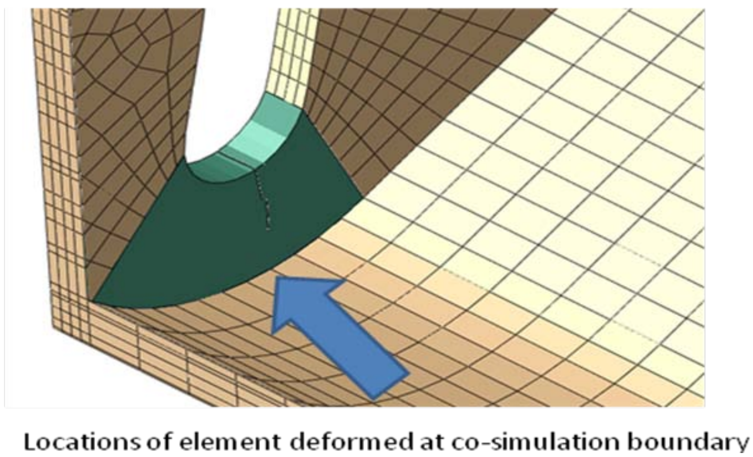

Figure 20-Deformed boundary elements

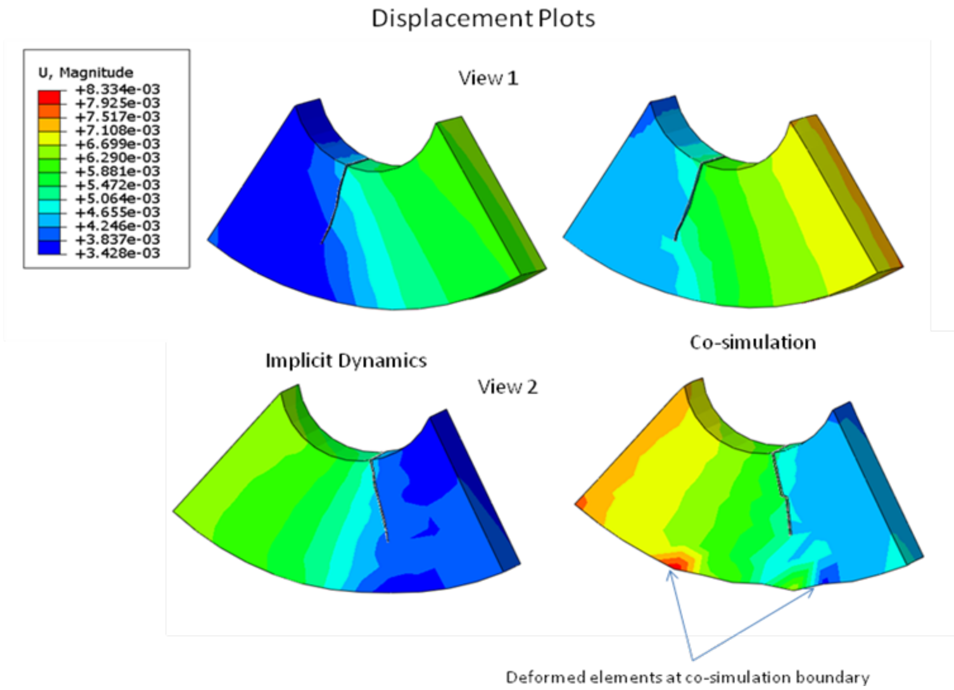

Figure 21-Displacement Plots (inches)

DISCUSSION

It was found that the following considerations improved the co-simulation analysis.

1. Use matching mesh's at the co-simulation boundaries. The co-simulation boundaries will miss nodes that are not nearly coincident, in that case no load will be applied to those nodes.

2. Start both analysis with the same initial time step. Starting both analysis at the same time improved the convergence of the implicit analysis.

3. When adding the XFEM enriched area, there is an additional requirement to keep the enriched area away from the co-simulation interaction boundary, to improve convergence and prevent cracks from starting at the co-simulation boundary, figure 11.

To validate the model, comparisons were made between a continuous mesh model and a tied mesh model to co-simulation model. Both the continuous and tied models were implicit dynamic analysis. In the co-simulation, the bulk of the model was an explicit dynamic analysis, only a small portion of the handle was run as a implicit dynamic analysis. The stress values matched well between the co-simulation and the tied implicit model (figure 15). There was a greater difference between the continuous mesh model and the other two models, but even this difference is small.

There was some difference in displacements between the co-simulation and tied parts (figure 19,21). This is expected as sub-cycling was allowed, this method matches velocities instead of displacements so some drift is expected. There is an issue with the distortion of some elements at the co-simulation boundary (figure 19,21). A clever means to eliminate this error was not found. Perhaps the error is caused by the edge of the co-simulation interaction being on a face that extends past it (figure 20). This may be the cause of the excessive element distortion.

Though the bulk crack shape matched well (figure 16,17), the crack initiation sites were different and the cracks initiated at slightly different times(figure 18). Eliminating the element distortion at the co-simulation interface and using the lockstep method might reduce this error.

In both cases the crack reached the edge of the enriched area. If the enriched area was extended crack growth probably would have continued.

Using co-simulation sped up the analysis considerably. Running on a Linux cluster the implicit dynamic analysis took 6 hours to run, the co-simulation took 10 minutes. The implicit dynamic analysis used 12 cpus, the co-simulation used 12 cpus for the explicit analysis and 1 for the implicit analysis.

CONCLUSIONS

In general, the co-simulation was successful. The stress results between the implicit dynamic and co-simulation analysis were a good match. After adding XFEM the run time was significantly reduced with the co-simulation compared to the implicit analysis. There was some variation in crack initiation location and time, between the two analysis, but the bulk crack growth matched very well. These results could probably be improved by the following:

1. Moving the co-simulation boundary farther from the area of interest.
2. Using the lockstep method
3. Moving the co-simulation interaction boundary so is does not share the extended face (figure 20).

REFERENCES

[1] L. Reinhardt, J.A., Cordes, “XFEM Modeling of Mixed-Mode Cracks in Thin Aluminum Panels”, Presentation at the 2010 Simulia Customer Conference, May 25-27 Providence, Rhode Island.

[2] “ABAQUS User Manual V6.9-EF1”, Dassault Systems 2004-2010.