Today, simulation has become an integral business practice across a wide spectrum of industries, whether you are building an airplane made of composites, designing a small hand-held electronic component, or developing a medical device. Predicting and designing against the failure of such products by fracture or fatigue is an essential, yet challenging part of the design and simulation process.

As SIMULIA’s technology leader for fracture & failure technology, my professional involvement in fracture mechanics and Abaqus can be traced back nearly 25 years. I first used Abaqus (Version 4) as a graduate student working for my Ph.D. on fracture of tubular joints in an offshore structure at the University of Glasgow, Scotland. I joined SIMULIA in 1996 and have been focused on our fracture and failure technology from the beginning. When I started, fracture and failure simulation was performed only by the researchers. It required very deep knowledge of fracture mechanics and numerical methods and was used by only a small subset of the Abaqus customer base.

Over the past 15 years, I have been one of the key contributors within our team of talented developers on advancing the fracture and failure technology in Abaqus. We have a well-defined R&D strategy in place to help our customers achieve their failure analysis objectives. Our long-term goal is to make the modeling of fracture and failure as common and as easy as including the effects of plasticity.

Customer-driven—industry-focused capabilities
To provide the best solutions to our customers, it is crucial that we clearly understand their fracture and failure requirements. To this end, we formed the SIMULIA Fracture Customer Review Team (FCRT) in 2003, which is composed of leading industry and academic experts. The FCRT meets annually with our developers and product managers to help identify and prioritize the development of new capabilities and validate the new functionality.

The input from this team, as well as other customers, has led directly to the release of substantial new fracture capabilities, including cohesive elements, Virtual Crack Closure Technique (VCCT), material damage models, J-integral contour modeling, low cycle fatigue with direct cyclic, and many others over the past eight years, beginning with the Abaqus 6.4 release. Additionally, a significant number of innovative capabilities were developed through customer-requested enhancements and direct engagements with our customers and partners. One such partnership, with Boeing Commercial Aircraft Group, enabled us to deliver the VCCT technique within Abaqus/Standard in version 6.5. This technology has made a positive impact on the use of composite materials in fabricated structures today.

Another R&D philosophy we’ve adopted is to pay close attention to applications in other industries when working with
customers on a primary industry application. One example is a cost-shared development project with PSA Peugeot Citroën to develop direct cyclic analysis technique on a cylinder head and exhaust manifold in the Automotive industry. The direct cyclic solution method is a unique capability in Abaqus to automatically calculate the stabilized, steady-state response of a structure to cyclic loading, thus increasing solution accuracy and reducing solution time compared to traditional methods. The direct cyclic analysis technique has also found many applications in Aerospace, High-Tech, and Life Science industries. In two IMECE papers, published in 2006 and 2007 respectively, Intel engineers described how they applied the direct cyclic solution method in Abaqus for thermal fatigue analysis of electronic packages.

By working closely with our customers and being cognizant of different industry needs, we are able to understand the various industry-specific processes and simulation requirements and align our development efforts towards solving real engineering problems.

Innovative technology for fracture & failure

In May 2009, the Abaqus 6.9 release continued to deliver our strategy with the introduction of a framework for applying the eXtended Finite Element Method (XFEM). This technology is an extension of the conventional finite element method based on the concept of partition of unity, which allows local enrichment functions to be easily incorporated into a finite element approximation. The presence of discontinuities is handled by the special enrichment functions in conjunction with additional degrees of freedom. A distinguishing feature of XFEM is that the model geometry does not have to conform to the crack—this leads to significant simplifications, especially when it comes to creating a model and simulation of crack propagation. In other words, with XFEM cracks can propagate in any direction across a mesh, even through existing elements.

XFEM, as a formalized approach, has been in the technical literature since the first paper was published by Belytschko and Black in 1999. It has gained acceptance in the academic world as a viable approach to modeling fracture. However, the state-of-the-art ideas found in the open literature are not general or robust enough to be used in a high-quality commercial product such as Abaqus. Many technical and architectural issues needed to be resolved by the SIMULIA XFEM development team. More importantly, Abaqus is a general purpose code and developers have to write a method that can be used for a multitude of physical problems. The challenge is perhaps at least as big as coming up with an idea that works for only a few specific problems in academia. It has to be robust, accurate, and easy-to-use. SIMULIA has recognized that completely developing the XFEM capability is a multi-year development team effort and has invested significant R&D resources into this endeavor over the years.

Since XFEM was first included in the Abaqus 6.9 release, we have rapidly added and enhanced the functionalities to support our customer’s growing interest in the solution. Such improvements include support for:

- Contour integral evaluations for an arbitrary stationary crack
- The implicit dynamic option for transient analysis
- The Linear Elastic Fracture Mechanics (LEFM) approach to complement the cohesive segments approach
- Low cycle fatigue analysis within the framework of direct cyclic analysis and user-defined damage initiation criteria
- Second-order tetrahedron elements to go mainstream with XFEM and contour integral evaluations with residual stress field.

Measuring success

At the 2011 SIMULIA Customer Conference (SCC), I moderated the Special Interest Group (SIG) on fracture and failure which was well-received by approximately 60 customers. The goals of the SIG are to interact more closely with attendees on industry-focused topics, provide value through close interaction with SIMULIA R&D, provide a review of our historical progress, current status, and a peek at our future strategy, and to allow feedback on our products, company, and strategy. Our customers are very energized by this topic and the fact that we have made excellent progress on delivering innovative applications of the XFEM technology in each release (with Abaqus 6.11 being the fifth release).
Strategy Overview

In this pressure vessel model, XFEM in Abaqus/Standard allows for the prediction of arbitrary, solution-dependent crack growth independent of the finite element mesh.

Our customers are particularly happy that our fracture and failure development is guided by the FCRT and some of them have expressed great interest in joining the team. At the same time, we greatly appreciate the feedback and the enhancement requests made during the Q&A.

The SCC provides an excellent forum to directly interact with many customers about fracture and failure. Out of 95 customer presentations, there were seven customer papers on XFEM, in addition to a half dozen on fracture and failure with conventional finite elements. These papers were on applications across a wide spectrum of industries, including Industrial Equipment (Tenaris Dalmine on pipe rupture), Energy (TÜV NORD on fracture of pressure vessel due to operational thermal loads using XFEM), Defense (U.S. Army–ARDEC on fracture of equipment, such as a hand grenade drop test or gun breech, using co-simulation in conjunction with XFEM), and Life Sciences (University of Iowa on ceramic total hip bearing fracture using XFEM). It is always good to see our customers putting Abaqus XFEM technology into action! It's evident that the efforts by the XFEM team over the past four years have certainly paid off.

**Future endeavors with fracture & failure technology**

The FCRT will continue to help SIMULIA align our strategy in fracture/failure/fatigue simulation with real industry needs, fill in any missing functionality in the first generation of enhancements, and prioritize work on these enhancements. We are fully committed to developing complete and robust XFEM functionality across a wide section of our customer base in all industries. Our customers are expected to see continued developments focused in the area of multiphysics simulation, as well as improvements in ease-of-use. Significant challenges remain to deploy the XFEM method to non-experts and to integrate this technology into the next generation of products built on Dassault Systemes Version 6 platforms. We are proud of what the FCRT has accomplished so far and excited about where XFEM, and all the other fracture and failure technology we have today, will lead us in the coming years.

**Zhen-zhong Du,**

Technology lead for fracture & failure technology

Zhen-zhong's professional involvement in fracture mechanics and Abaqus can be traced back nearly 25 years. He first used Abaqus as a graduate student working for his PhD on fracture mechanics at the University of Glasgow, Scotland. Working with his advisor, he co-authored a paper on constraint effects that was published in the Journal of Mechanics and Physics of Solids* in 1991. He then worked at Cambridge University and at the University of California at Santa Barbara prior to joining SIMULIA in 1996. He has been one of the key contributors within SIMULIA's talented team of developers on advancing the fracture and failure technology in Abaqus.


For More Information

www.simulia.com/XFEM