Ansys

WHITE PAPER

SMART Fracture

According to a 1983 report by Battelle and the National Bureau of Standards, the economic costs of fracture in the United States alone was $119 billion per year (in 1983 dollars). If a similar study were to be done today, that figure would surely have increased, given the large growth in consumer products, automobiles and industrial equipment over the years. Any technology that could reduce the costs of fracture to consumers and businesses could have a big impact on the economy, as well as customer satisfaction.

The traditional fracture testing method of building prototypes is a time-consuming and costly process. Engineering simulation of fracture during the design phase to predict a product's toughness has been an available alternative method since the early 1980s. However, simulation of fracture has not been easy to set up and perform until now. Traditionally, the meshing stage consumed a lot of time (up to several days) because engineers had to fit a crafted mesh involving hexahedrons (hex) and wedges to capture the crack front — the ideal mesh using only hex elements was often not possible. This led engineers to simplify the geometry of the crack to fit a hex-only mesh pattern, resulting in loss of fidelity in fracture analysis.

Now, with the new Unstructured Mesh Method (UMM) in Ansys Mechanical, engineers can reduce preprocessing time by employing UMM's automatically generated all-tetrahedral (tet) mesh for crack fronts, while achieving the same high-fidelity results as a simulation run with the ideal hex mesh configuration. Meshing time has been reduced from up to several days to a few minutes.

Using UMM, Ansys has also introduced the Separating Morphing and Adaptive Remeshing Technology (SMART) crack growth simulation technology to Ansys Mechanical to enable automatic remeshing during a simulation. A SMART simulation can be set up with several clicks, eliminating long preprocessing sessions.

/ Fracture Analysis Parameters and Methods

It is difficult to produce a perfect material using traditional manufacturing methods or additive manufacturing. Imperfections resulting from inclusions, grain boundary mismatches, differential thermal expansion, contamination of the melt or many other mechanisms can lead to imperfections.

These imperfections can be the starting points for cracks. Depending on the size and shape of the crack and the conditions under which the component is used, a crack could be stable and cause no problems at all, or unstable, in which case it will grow at a rate that depends on the forces acting on it and the fracture toughness of the material in which it resides. Cyclic loading can initiate a crack in a structurally weak area of a component; the crack can then grow due to fatigue. The growth of a crack throughout the volume of a component can result in catastrophic fracture.

The size of a flaw in any material is an important variable in fracture analysis, as is the stress on the flaw and the fracture toughness of the material, which replaces material strength in fracture calculations. The stress intensity factor (SIF) determines the fracture toughness of a material subject to linear-elastic fracture mechanics (LEFM); the SIF variable is represented as K_{IC}. For elastic-plastic fracture mechanics (EPFM), fracture toughness is determined by the energy required to grow a crack, represented by J_{IC}, also known as the J-integral parameter. Fatigue crack growth has been modeled using Paris' law:

$$\frac{da}{dN} = A(\Delta K)^B$$

Where A and B are material dependent constants.

Many other parameters have been defined over the years to describe crack growth in various conditions or types of materials.

Ansys has recently added the Material Force parameter to this list. Material Force is a generalized parameter that supports various linear and nonlinear materials. It enables engineers to use a single parameter to characterize different material models. While Material Force will become an increasingly valuable parameter in fracture analysis in the coming years, Ansys Mechanical will continue to support SIF and J-Integral analyses also.

Another consideration is the mode of fracture. Three modes are typically recognized: Mode I, or tensile mode, in which the forces are perpendicular to the crack; Mode II for shearing; and Mode III for out-of-plane tearing.

A mixture of these modes is also possible. In this paper, we will limit our discussion to Mode I crack propagation.

/ Crack Simulation Methods

Fracture simulation has relied on two models: traditional cohesive zone modeling (CZM) and, more recently, the eXtended finite element method (XFEM).

CZM is mostly used for simulating debonding between two surfaces attached adhesively. Delamination occurs when a load is applied, but the delaminating crack cannot grow beyond the interface of the two surfaces. CZM is valuable for simulating composites, but it is generally not suitable for simulating a crack growing in the bulk of a material.

XFEM is better for internal crack calculations. Introduced into the Ansys toolkit a few years ago, XFEM eliminates the need of remeshing crack tip regions. Instead, it defines an extended finite element enrichment area around a crack tip and in regions where it is plausible that the crack tip might grow. XFEM splits the special volume elements in the enrichment zone from the center of the element. In this way it creates a finer mesh by splitting existing cells instead of remeshing.

But the enrichment regions in XFEM are very computationally expensive. As the enrichment area increases, the simulation slows down. So XFEM is not easy to scale up to large projects. XFEM results can also be influenced by the underlying mesh.

/ Introducing Ansys SMART Crack Growth with UMM

The latest innovation by Ansys is Separating Morphing and Adaptive Remeshing Technology (SMART), which relies on the UMM process introduced above.

UMM is more versatile and easy to use than any previous fracture simula- tion technology. Engineers can place a crack of any shape at any location within the geometry of the part being simulated, including at the surface or in the bulk, and use simulation to determine the rate of crack growth (if any) and the number of cycles to failure in the case of fatigue conditions. Remeshing with tet elements is done automatically at a critical region around the crack tip at each iteration of the simulation process. Automatic remeshing refines calculations in the most needed regions without requiring the engineer's intervention. All of this is accomplished without compromising or influencing the accuracy of the results.

In one example, engineers simulated a turbine blade with a semi-elliptical crack near the hub. When UMM was turned off, the SIF stress intensity factor fluctuated greatly – sometimes exceeding the level of accuracy required. With UMM turned on, the same simulation matched the results of one performed using an ideal hex mesh.

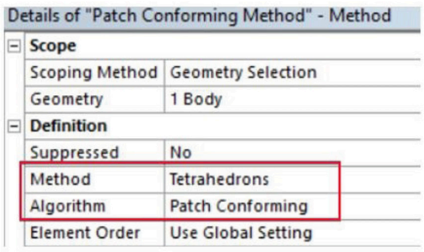
Before UMM, engineers had to define the crack front location along with its major and minor axes, even for ideal semi-elliptical cracks. Now, UMM can handle the arbitrary cracks that are more generally found in real applications. Engineers can even perform an ultrasound scan of the component to find the true shape of the crack in a sample, extract that shape and put it into the model to see how the crack behaves.

SMART updates the mesh from crack-geometry changes due to crack growth automatically at each solution step instead of using enrichment area (splitting) of XFEM. Unlike XFEM, SMART can be scaled up for larger projects because remeshing is limited to a small area around the crack tip at each iteration. A big advantage of SMART is that you can use standard elements already included in Mechanical, with no need to develop new elements.

Engineers may also be able to enter any crack growth law into the software, perhaps a law proposed in a journal paper about the material being used.

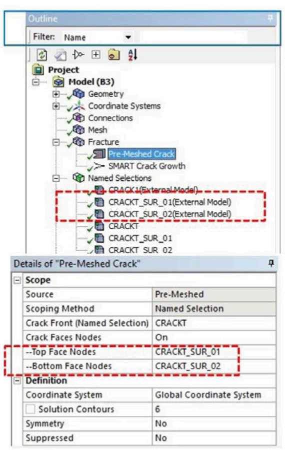
Setting Up and Running an Ansys SMART Simulation

Setting up a SMART simulation can be accomplished with just a few clicks. After importing the geometry, a mesh method needs to be added and set to 'Tetrahedrons'; by default the Patch Conforming algorithm will be chosen:

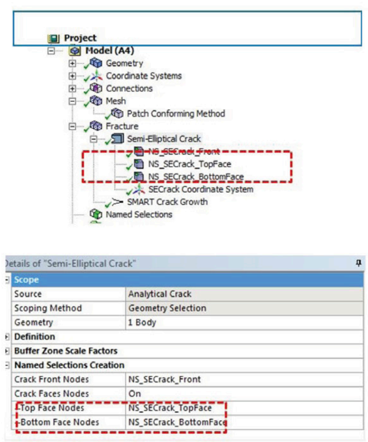


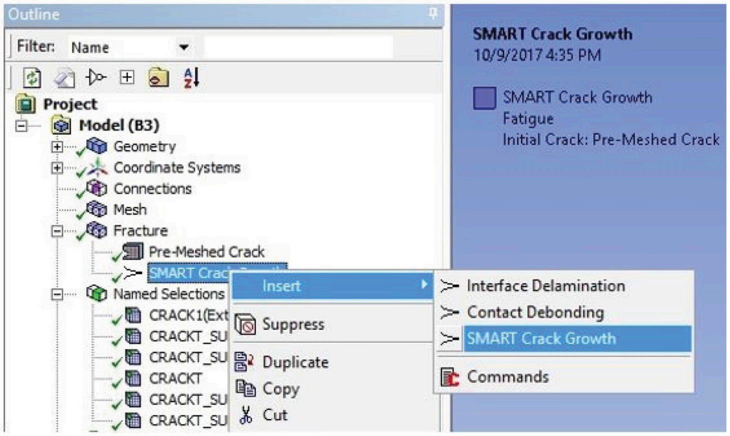
The next step is initial crack definition. Crack Faces Nodes need to be set to 'On' for SMART crack-growth.

PRE-MESHED CRACK



SEMI-ELLIPTICAL AND ARBITRARY CRACK





Details of "SMART Crack Growth"

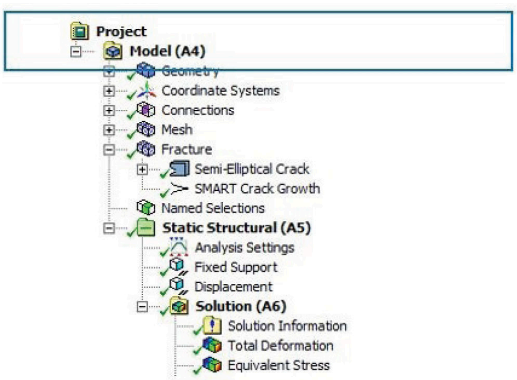
Definition	
Analysis	Crack Growth
Method	SMART
Suppressed	No
Options for Crack Growth	
Initial Crack	Pre-Meshed Crack
Crack Growth Option	Fatigue
Failure Criteria Option	Material Data Table
Material	Structural Steel
Crack Growth Law	Paris Law
Crack Growth Methodology	Life Cycle Prediction
Min Increment of Crack Extension	Manual
--Min Increment Value	1.e-003 m
Max Increment of Crack Extension	Manual
--Max Increment Value	0.25 m
Stress To Load Ratio	0.9

It could be Pre-Meshed,
Semi-elliptical or Arbitrary crack.

Crack growth option: Static or Fatigue.

Crack growth Methodology:
Life cycle pediction or Cycle by cycle.

Set up and solution:

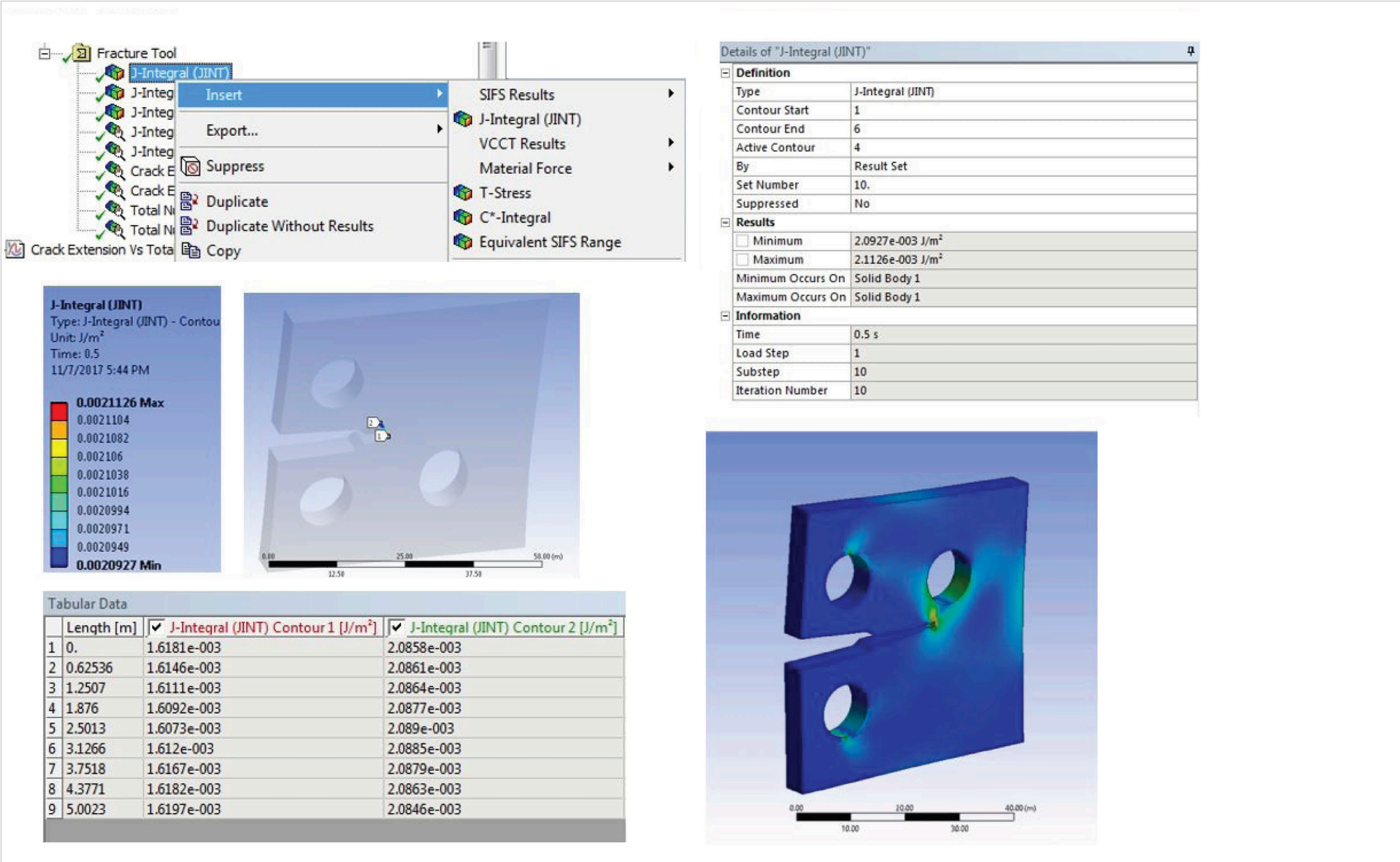


Details of "Analysis Settings"

Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Off
Define By	Substeps
Number Of Substeps	4.
Solver Controls	
Rotordynamics Controls	
Restart Controls	
Fracture Controls	
Fracture	On
SIFS	Yes
J-Integral	No
Material Force	No
T-Stress	No
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Visibility	

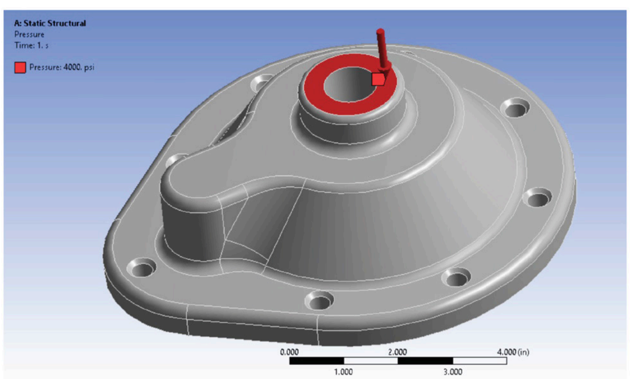
```
 CINT CALCULATION FOR CRACK #  1
      NUMBER OF CONTOURS        6
      SYMMETRICAL CRACK       NO
      CRACK PLANE NORMAL      CSYS= 13 AXIS= 2
      UMM KEY IS              ON

 CRACK GROWTH CALCULATION #    1
      CRACK ID              #    1
      SMART LOCAL REMESHING ATTEMPT # 1    SUCCESSFUL
      TOTAL ACCUMULATED MAX CRACK EXTENSION INC     2.0288

'** LOAD STEP     1   SUBSTEP     1  COMPLETED.    CUM ITER =      1
'** TIME =   0.250000         TIME INC =   0.250000      NEW TRIANG MATRIX

IAKE COMPONENTS FOR CRACK DATA SET     1

 CINT CALCULATION FOR CRACK #  1
      NUMBER OF CONTOURS        6
      SYMMETRICAL CRACK       NO
      CRACK PLANE NORMAL      CSYS= 13 AXIS= 2
      UMM KEY IS              ON

 CRACK GROWTH CALCULATION #    1
      CRACK ID              #    1
      SMART LOCAL REMESHING ATTEMPT # 1    SUCCESSFUL
      TOTAL ACCUMULATED MAX CRACK EXTENSION INC     3.9724

'** LOAD STEP     1   SUBSTEP     2  COMPLETED.    CUM ITER =      2
'** TIME =   0.500000         TIME INC =   0.250000      NEW TRIANG MATRIX
```

Simulation crack growth and fracture results:



	Length [m]	J-Integral (JINT) Contour 1 [J/m²]	J-Integral (JINT) Contour 2 [J/m²]
1	0.	1.6181e-003	2.0858e-003
2	0.62536	1.6146e-003	2.0861e-003
3	1.2507	1.6111e-003	2.0864e-003
4	1.876	1.6092e-003	2.0877e-003
5	2.5013	1.6073e-003	2.089e-003
6	3.1266	1.612e-003	2.0885e-003
7	3.7518	1.6167e-003	2.0879e-003
8	4.3771	1.6182e-003	2.0863e-003
9	5.0023	1.6197e-003	2.0846e-003

Here we simulate a surface crack inside a pump housing. The pump housing is subject to a load of 4,000 psi at the top surface and it is fixed at the bottom surface.

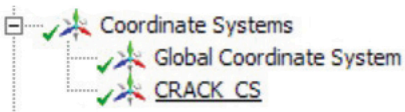


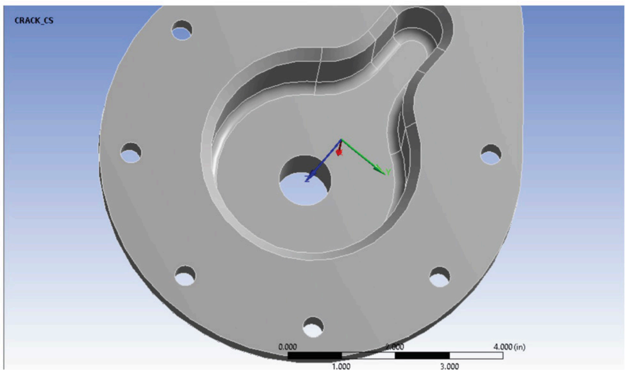
On the inside surface of this pump housing, it is discovered that there are elliptical/semicircular surface cracks left over from either machining or from field service. The objective of the engineer is to determine what size cracks will propagate and under what loading conditions so that the engineer can notify field personnel to remove and replace the service parts with a certain crack size. To be able to come up with an engineering simulation and report like this used to take many weeks in the past. But, using Ansys Workbench, the same report can be put together in a matter of hours given the current Ansys fracture mechanics capabilities.

The steps can be summarized as follows:

- Insert “Fracture” in the Mechanical application.

- Insert a local coordinate system on the surface where the crack is located.





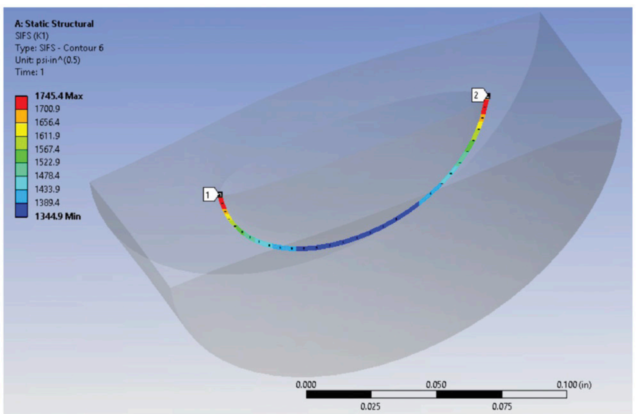
- Specify crack geometry in the lower left-hand corner.

Scope	
Source	Analytical Crack
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Coordinate System	CRACK_CS
Align with Face Normal	Yes
Project to Nearest Surface	Yes
Crack Shape	Semi-Elliptical
--Major Radius	5.e-002 in
--Minor Radius	5.e-002 in
Mesh Method	Hex Dominant
Largest Contour Radius	2.5e-002 in
Crack Front Divisions	15
Fracture Affected Zone	Program Controlled
Fracture Affected Zone Height	6.5035e-002 in
Circumferential Divisions	8
Mesh Contours	6
Solution Contours	Match Mesh Contours
Suppressed	No

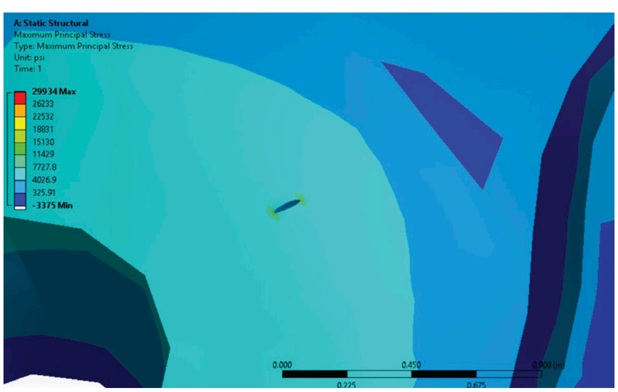
- Mesh with the new crack.
- Run the simulation.
- In the “Solution” section, insert “Fracture Tool”.

Details of "Fracture Tool"	
Scope	
Scoping Method	Crack Selection
Crack Selection	Semi-Elliptical Crack
Definition	
Suppressed	No

- Post process (extract) “SIFS (KI): KI, the “Mode I Stress Intensity Factor,” is very important in determining if the crack will experience either “instantaneous fracture” or if it will “propagate”. If K1 is less than K_{IC}, which is a material property, the critical stress intensity factor, then there is no immediate fracture. The next step then is to compare K1 with “Threshold Stress Intensity Factor Kth”. In most cases, Kth is of the order of 2 ksi·$in^{0.5}$. In this example, K1 is 1.7 ksi $in^{0.5}$, therefore this crack will not propagate under this load and at this location; a 0.05 inch semicircular crack can be tolerated and still kept in service for this case.



- The next step for the engineer is to change the:
  - Location of the crack.
  - Size of the crack.
  - Load (pressure) upwards (increasing pressure).
  - Displacement boundary conditions.



Summary

SMART simulation is the latest in a long line of Ansys innovations designed to solve the critical issue of crack initiation, growth and fracture in product design. Remeshing around the crack tip after each iteration concentrates the computing power where it is needed most, making SMART simulations faster and easier to scale up for larger projects. Using UMM for meshing eliminates costly preprocessing time and produces a tet mesh that is just as accurate as a time-consuming hex mesh. Ansys innovations continue to save you time and money while helping you get your reliable products to market faster.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com