Advances in Simulation of Arbitrary 3D Crack Growth using FRANC3Dv5

Wawrzynek, P. A.[*]　Carter, B. J.[*]　Hwang, Changyu[†]　Ingraffea, A. R.[**]

Abstract

FRANC3D is a program for simulating arbitrary three-dimensional crack growth. Recently, a completely new version of the program, FRANC3D/NG, has been created. Unlike previous versions, which relied largely on boundary element analysis, the new version of the program works with finite element analysis exclusively and is designed to work with general-purpose commercial finite element packages. This paper presents the theoretical underpinnings of the procedures to adaptively modify the geometry and mesh of a model to simulate crack growth.

Keywords : *crack growth simulation, stress intensity factors, finite element analysis, remeshing*

1. Introduction

The FRacture ANalysis Code 3D Version 5 (FRANC 3Dv5 for short) is designed to simulate crack growth in engineering structures where the component geometry, local loading conditions, and the evolutionary crack geometry can be arbitrarily complex. It is designed to be used as a companion to a general purpose Finite Element (FE) package. Currently, interfaces to the ANSYS, ABAQUS, and NASTRAN commercial programs are supported. FRANC3Dv5 is a successor to the original FRANC3D program, which was developed at Cornell University in the late 1980′s. While the two codes share a name, version 5 is a complete rewrite of the code. It employs different approaches for geometrical modeling and deformation analysis and benefits from over 20 years of experience developing and using earlier versions of the code. The typical workflow for a FRANC3Dv5 analysis is shown in Fig. 1. An analyst creates an uncracked FE mesh using the standard tools available for the commercial FE package. Typically, the analyst then defines a sub-model of the crack growth region. FRANC3Dv5 reads the sub-model mesh file and remeshes the sub-model to incorporate the geometry of a crack. The crack geometry and location can be prescribed either interactively using the Graphical User Interface (GUI) or programmatically using FRANC3Dv5 extensions to the Python programming language. The “cracked” sub-model is reintegrated into the remainder of the model and an analysis is performed. The resulting

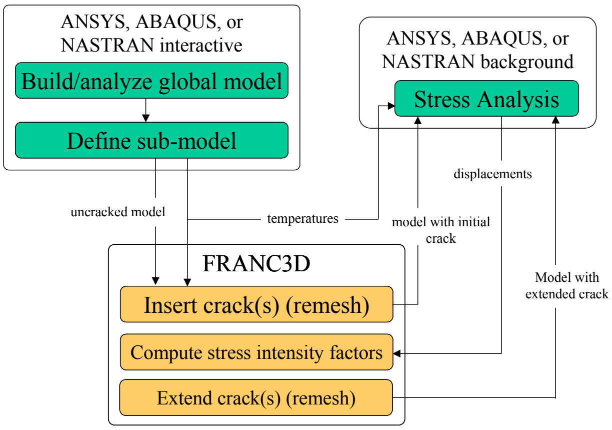

Fig. 1 Typical workflow for a FRANC3Dv5 crack growth analysis

---

[†] 책임저자, 정회원・서울벤처정보대학원대학교 U-City학과 조교수
Tel: 010-6416-0053 ; Fax: 02-523-6767
E-mail: hwang@suv.ac.kr
[*] Fracture Analysis Consultants, Inc., Ithaca, NY, USA
[**] Cornell University, Ithaca, NY, USA

•이 논문에 대한 토론을 2011년 2월 28일까지 본 학회에 보내주시면 2011년 4월호에 그 결과를 게재하겠습니다.

displacements are read back into FRANC3Dv5, which then computes Stress Intensity Factors (SIF's) for all node points along the crack front. The SIF's are used to predict the direction and relative amount of growth of the crack front points. The crack is extended, the sub-model remeshed, and another stress analysis is performed. This process is repeated for the number of crack steps specified by the analyst. This paper is organized into five sections. The following section describes aspects of the crack growth modeling approach used within FRANC3Dv5. The third section is a brief overview of FRANC3Dv5's geometrical modeling and meshing pipeline. This is followed by an illustrative example and a brief summary.

2. Modeling Approach

In this section, key aspects of how FRANC3Dv5 operates and interacts with other programs are described. FRANC3Dv5 supports the notion of a sub-model; remeshing for crack growth is confined to this sub-model. The sub-model approach is illustrated in Fig. 2. All the supported FE packages have tools to define a sub-model. In a "real world" analysis, the size of a crack is small relative to the size of the structure. Confining the remeshing for crack growth to the sub-model greatly reduces the amount of data that needs to be transferred to, and processed by, FRANC3Dv5, thus speeding the crack growth process. It also allows the analyst to leave intact portions of a model with different structural idealizations (e.g. shell elements), complex boundary conditions (e.g., contact), or naturally and easily meshed with brick elements (FRANC3Dv5 remeshes with predominantly tetrahedral elements, as described below). Sub-modeling is used for mesh modification only; it does not affect the analysis strategy. That is, the remeshed sub-model is "plugged" back into the global model and the stress and deformation analysis is performed for the full composite model. This approach is not a sub-structuring or local/global analysis methodology. The sub-model can be redefined at any step of a crack growth analysis. The main form of communication between FRANC3Dv5 and FE programs is through ASCII mesh description files. These are human-readable files that define a mesh, node coordinates, materials, and boundary conditions. These files use proprietary formats defined by the FE package vendors (.cdb, .inp, and .bnf formats for ANSYS, ABAQUS, and NASTRAN, respectively). FRANC3Dv5 expects a mesh file as input that describes an uncracked model or sub-model. It may also request a file of nodal temperatures in the case of thermal/ mechanical loading, and possibly a file of nodal stresses if initial or residual stress fields are to be considered The output from the program is a new file that describes a mesh for a model or sub-model containing a newly inserted or newly extended crack (and possibly a file of interpolated nodal temperatures). Transferring a mesh description of a component between FRANC3Dv5 and a FE program is sub-optimal because mesh descriptions encode geometrical information incompletely. For example, if a portion of the surface of a component is curved, then the mesh model will have replaced that surface with a collection of planar or polynomial patches. This means that FRANC3Dv5 must use heuristic algorithms to reconstruct a description of the local geometry from the mesh data (a procedure described below). In most cases, the reconstructed geometry will be approximate. In theory, a better approach would be for the FE package to send FRANC3Dv5 geometrical data and the mesh data. In practice, however, this would introduce unwanted complexities. The format of mesh files varies among

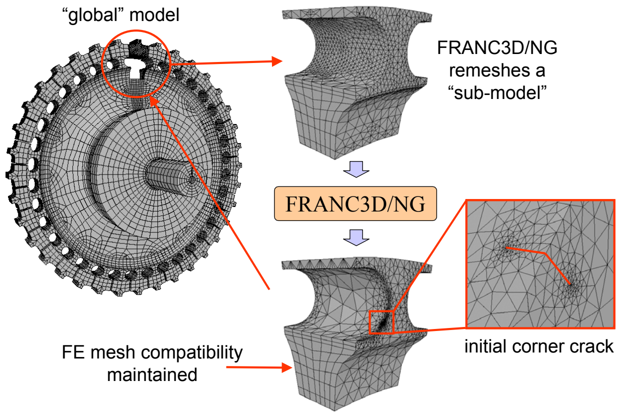

Fig. 2 The sub-modeling approached used in FRANC3Dv5

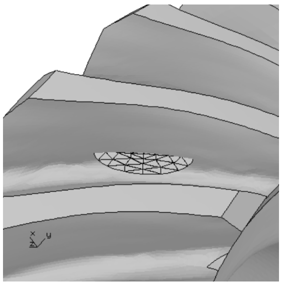

Fig. 3 An image from the flaw insertion wizard showing an elliptically shaped crack which is positioned in a gear tooth

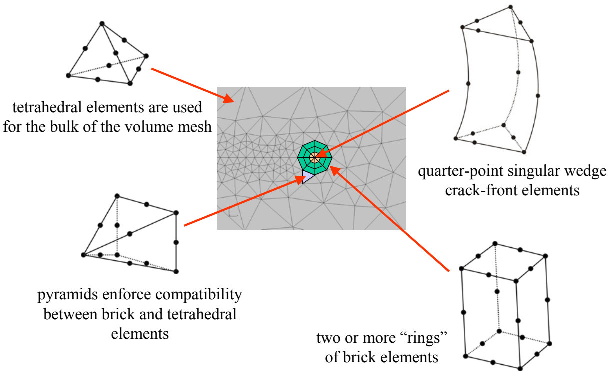

Fig. 4 Element types used to mesh the crack-front region

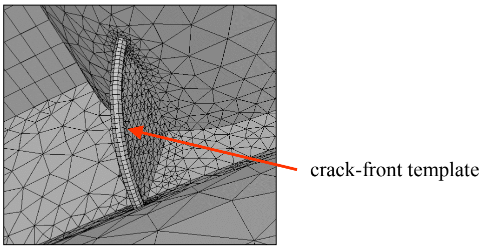

Fig. 5 A crack-front template

venders, but the main information they contain (node coordinates and element descriptions) is essentially the same among popular FE packages, and they describe a relatively simple data model. True geometrical information, in general, is much more complex and can vary markedly among vendors. Frequently, an analyst tasked with performing a crack growth investigation does not have easy access to a solid model description of a component and can more easily work with a mesh model. On balance, even though exchanging FE data introduces geometrical approximations, doing so makes FRANC3Dv5 a more flexible and useful tool than if it were tied to the geometrical information demanded by a specific FE package. FRANC3Dv5 can be used to insert both zero volume flaws (cracks) and finite volume flaws (voids) into a model. Both types of flaws are defined as a collection of triangular cubic Bézier spline patches. Using spline patches to describe flaws means that very complex, doubly curved crack surfaces can be modeled. In most cases, however, an analyst will start by inserting an initial flaw with a relatively simple geometry and have it grow into a more complex shape. FRANC3Dv5 provides a “wizard” to specify an initial flaw shape, orientation, and location. One first selects from a small library of parameterized flaw shapes (e.g. elliptical crack, part-through crack, center crack, ellipsoidal flaw). Then one specifies translations and rotations to position the flaw. The wizard provides visual feedback so that one can confirm the flaw is in the proper location. An image crack insertion wizard is shown in Fig. 3.

Notice in Fig. 3 that the wizard allows the analyst to work with the full elliptical crack shape, even though only about half the ellipse will ultimately be inserted into the part. The program computes the intersection of the flaw with the (in this case, doubly curved) surface and trims the unused portion of the flaw. This means the same parameterized elliptical crack model can be used to define a wide range of fully embedded, surface, or corner crack geometries. A variety of element types are used within FRANC3Dv5 for meshing near cracks. As illustrated in Fig. 4, 15-nodes wedge elements are used adjacent to a crack front. By default, eight wedge elements are used circumferentially around the crack front and these elements have the appropriate side-nodes moved to the quarter points, which allows the element to reproduce the theoretical 1/r stress distribution. The crack-front elements are surrounded by “rings” of 20-noded brick elements (two rings by default). Together, the wedge and brick elements comprise what is referred to as the crack front “template”. The template is extruded along the crack front a shown in Fig. 5. This regular pattern of elements in the template is exploited when computing conservative integrals (e.g., J-integral (Rice, 1968) and M-integral (Yau *et al.*, 1980; Yau and Wang, 1984)).

The bulk of the sub-model is meshed with 10-noded tetrahedral elements. The triangular faces of the tetrahedral elements are not compatible with the quadrilateral faces of the brick elements; 13-noded pyramid elements are used to transition from the template to the tetrahedra. Not all finite element packages support pyramid elements, so as an option the pyramid elements can be divided into two tetrahedra with the "hanging" node constrained. FRANC3Dv5 computes stress intensity factors using an M-integral approach. The M-integral, sometimes called the interaction integral, computes the energy release rates segregated by modes so the modal SIF′s (KI, KII, and KIII) can be computed. Computationally, the M-integral is similar to the J-integral (Rice, 1968). FRANC3Dv5 has M-integral formulations for both isotropic and orthotropic materials where the material axes can be oriented arbitrarily relative to the crack front (Banks-Sills *et al*., 2007; Yau *et al*., 1980; Yau and Wang, 1984). Within FRANC3Dv5, crack growth is a five-step process:

1) SIF's are computed for all node points along the crack front.
2) At each such point the direction and extent of growth is determined.
3) A space curve is fit through the new crack-front points and, for the case of a surface crack, extrapolated, if necessary, to extend outside of the body.
4) New Bézier patches are added to the crack surfaces.
5) The extended crack is inserted into an uncracked mesh.

By default, the Maximum Tensile Stress (MTS) criterion is used to predict the local direction of crack growth (Erdogan and Sih, 1963). Other direction criteria are available. The relative amount of crack growth for points along the front, by default, is a ratio of the corresponding SIF′s raised to an analyst specified power. This is analogous to evaluating the Paris crack growth rate equation for two points where both points are subjected to the same number of load cycles. In addition to a Paris type model, the NASGRO (NASA's standard program for fracture and fatigue calculations) or an analyst supplied (piecewise linear in log/log space) crack growth rate equation can be used to determine relative amounts of growth.

For more realistic 3-D fracture analysis, a virtual crack extension method for calculation of derivatives of energy release rates has been developed in conjunction with FRANC3Dv5 (Hwang and Ingraffea, 2007) while energy release rates and SIF are being calculated by M-Integral at the present version. The derivatives of energy release rate will provide useful information for the prediction of stability and arrest of a single crack, the growth pattern analysis of a system of interacting cracks, configurational stability analysis of evolving cracks, probabilistic fracture mechanics analysis and universal size effect model. In the case of multiple crack systems, for example, the variation of energy release rate at one crack tip due to the growth of any other crack must be calculated to determine the strength of the interaction. In probabilistic fracture mechanics analysis of linear-elastic cracked structures, the first and second order reliability methods require accurate estimates of energy release rates, their first and second order derivatives. Another use of the higher order derivatives is for size effect models that relate nominal strength to the structure size. The universal size effect model by Bazant requires the first and second order derivatives of energy release rate (Bazant, 1995).

3. The FRANC3Dv5 Geometry/Meshing Pipeline

The core of the FRANC3Dv5 program is the geometrical modeling and meshing pipeline that adapts a FE mesh to insert a crack. This is a four-step process: surface facets and edge detection, geometry reconstruction, intersection computations, and meshing. These steps are described briefly in the following paragraphs. As mentioned above, FRANC 3Dv5 takes as input a mesh description of model. The first step in the pipeline is to determine the FE facets that either make up the outer boundary of the sub-model or fall on a bi-material interface. Depending on

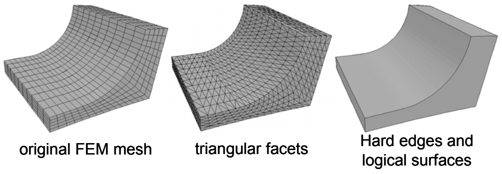

Fig. 6 Preliminary surface and edge detection

the solid element type, the surface facets will have either a triangular or quadrilateral shape. Quadrilateral element facets are further divided into two triangular facets yielding a planar-faceted, approximate geometrical description of the sub-model external and bi-material interface surfaces. The next step is to determine collections of the surface facets that can be grouped to form planar or gently curving regions that can be logically seen as a single surface, figure 6. The logical surfaces are bounded by "hard" edges. These are patch boundaries that one would perceive as making up a portion of a "sharp" edge of the model.

The primary heuristic used to identify sharp edges is to examine the angle between two adjacent surface patches. If the angle is below a threshold, the common boundary segment is flagged as a hard edge. Once all edges have been identified, strings of adjacent edges are determined and grouped to form circuits that form boundaries of logical surfaces. Exceptions to the patch grouping procedure are surfaces that fall on the interface between the sub-model and global model. These surfaces are retained as distinct patches so that nodal compatibility can be maintained when the sub-model is reinserted into the global mesh. The FE facets give an approximate description of the surface geometry of the sub-model. A more accurate, but still approximate, geometry description is obtained by replacing the planar triangular facets with triangular cubic Bézier patches, which can represent doubly curved surfaces. At this stage of the pipeline, both the component and the crack are represented as collections of triangular cubic Bézier patches. To "insert" the crack into a component, a search is made to find all intersections between crack and component patches (Bajaj *et al.*, 1988; Müllenheim, 1991). For intersected crack patches, those parts of the patch inside the component and those outside are determined. The external parts are "trimmed", and the internal parts are added to the component model to form a composite crack/component model. The intersection curves, which are the crack mouth, become additional hard edges in the model. The final stage in the pipeline is generating a FE mesh. Meshing is performed in four steps. First, FE nodes are generated along all hard edges. Second, triangular surface meshes are generated for all logical surfaces. Third, pyramid elements are generated adjacent to all quadrilateral boundary facets. Finally, a tetrahedral volume mesh is generated for the full sub-model. An advancing front meshing algorithm is used to generate surfaces meshes. One of two forms of this algorithm is used, depending on the geometry of the surface. If the surface is planar or nearly so, the surface boundary is mapped to a least squares fit plane, a 2-D mesh is generated in this plane, and the internally generated node points are mapped back so that they fall on the appropriate Bézier patches. If the surface has significant curvature, then a computationally more expensive advancing front procedure is used where all the computations and intersection checks are performed using the 3-D surface geometry. An advancing front meshing algorithm is used for volume meshing also. In this case tetrahedral elements are generated.

4. Example

For a simple demonstration analysis, a "half-penny" surface crack has been inserted into a thick-walled cylinder subjected to combined tension and torsion loading. The left image of Fig. 7 shows the FE mesh with a high concentration of elements near the crack mouth. The center image shows a detail of the mesh on the surface of the crack. The right image shows the crack surface after one step of crack growth. The crack extension is non-planar due to the mixed-mode loading. Fig. 8 shows a sequence of "cut-away" images illustrating the evolving crack geometry. The penultimate image shows that the crack front is automatically split into two fronts as the crack grows to the inner bore. The final image shows the final trace of

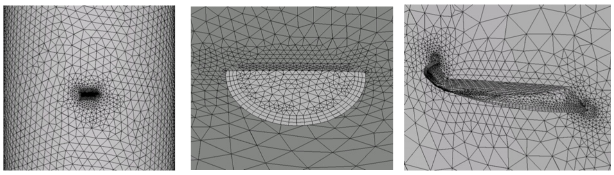

Fig. 7 Crack growth in a thick-walled cylinder: initial mesh (left), initial crack surface mesh (middle), crack surface mesh after one step of growth (right)

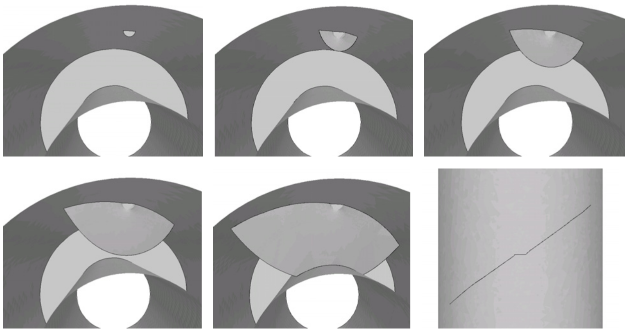

Fig. 8 The geometry of the evolving non-planar crack geometry

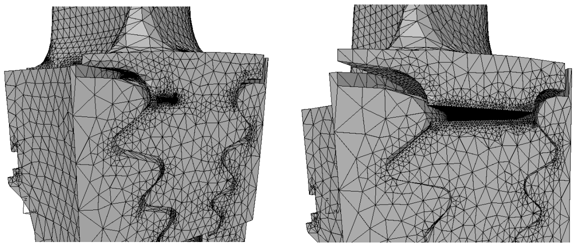

Fig. 9 Crack growth in the fir-tree attachment region of a turbine blade

the crack mouth on the surface of the cylinder.

Fig. 9 shows another example of crack growth in the fir-tree attachment region of a turbine blade.

5. Summary

FRANC3Dv5 is a program designed to simulate crack growth in engineering structures were the component geometry, local loading conditions, and the evolutionary crack geometry can be arbitrarily complex. It is designed to be used as a companion program to a general purpose FE package. Because stress analysis is performed by capable commercial packages, complex mechanics, such as contact and advanced material models, can be included in crack growth analyses. This paper has provided a brief overview of FRANC3Dv5. Because of length restrictions many aspects of the program have been treated very superficially. Little mention has been made of the graphical user interface, which allows an analyst to be productive with a shallow learning curve. The paper does show, however, how geometrical modeling, computational fracture mechanics, and meshing capabilities have been combined to create a tool that provides an analyst with the capability to model realistically shaped cracks in real engineering structures subjected to realistic loads.

Acknowledgements

We gratefully acknowledge that the development of FRANC3Dv5 was partially supported by the US Air Force (contract numbers FA6650-04-C-5212 and FA8650-07-C-5216) and the US Navy (contract number N68335-08-C-0011).

References

Bajaj, C., Hoffmann, C., Lynch, R., Hopcroft, J. (1988) Tracing Surface Intersections, *Comp Aided Geometric Design*, 5, pp.285~307.

Banks-Sills, L., Wawrzynek, P., Carter, B., Ingraffea, A., Hershkovitz, I. (2007) Methods for Computing Stress Intensity Factors in Anisotropic Geometries: Part II-arbitrary Geometry, *Engineering Fracture Mechanics*, 74(8), pp.1293~1307.

Bazant Z.P. (1995) Scaling Theories for Quasi-Brittle Fracture: Recent Advances and New Directions. In: Wittmann Folker H, editor. Fracture Mechanics of Concrete Structures, Proceedings FRAMCOS-2. D-79014 Freiburg: AEDIFICATIO Publishers; pp.515~34.

Erdogan, F., Sih, G.C. (1963) On the Crack Extension in Plates under Plane Loading and Transverse Shear, *Journal of Basic Engineering*, 85, pp.519~527.

Hwang, C.G., Ingraffea, A.R. (2007) Virtual Crack Extension Method for Calculating the Second Order Derivatives of Energy Release Rates for Multiply Cracked Systems, *Engineering Fracture Mechanics*, 74, pp.1468 ~1487.

Müllenheim, G. (1991) On Determining Start Points for a Surface/Surface Intersection Algorithm, *Comp*

Aided Geometric Design, 8, pp.401~408.

Rice, J.R. (1968) A Path Independent Integral and the Approximate Analysis of Strain Concentration by Notches and Cracks, *Journal of Applied Mechanics*, 35, pp.379~386.

Yau, J.F., Wang, S.S. (1984) An Analysis of Interface Cracks Between Dissimilar Isotropic Materials using Conservation Integrals in Elasticity, *Engineering Fracture Mechanics*, 20, pp.423~432.

Yau, J.F., Wang, S.S., Corten, H.T. (1980) A Mixed-Mode Crack Analysis of Isotropic Solids using Conservation Laws of Elasticity, *Journal of Applied Mechanics*, 47, pp.335~341.

- 논문접수일 2010년 10월 24일
- 논문심사일
  1차 2010년 11월 4일
  2차 2010년 12월 3일
- 게재확정일 2010년 12월 7일