1996-08-01

The Development and Evaluation of the knife Finite Element

Brett W. Clark
Brigham Young University - Provo

Follow this and additional works at: https://scholarsarchive.byu.edu/etd

Part of the Mechanical Engineering Commons

BYU ScholarsArchive Citation
https://scholarsarchive.byu.edu/etd/3456

This Thesis is brought to you for free and open access by BYU ScholarsArchive. It has been accepted for inclusion in All Theses and Dissertations by an authorized administrator of BYU ScholarsArchive. For more information, please contact scholarsarchive@byu.edu, ellen_amatangelo@byu.edu.
THE DEVELOPMENT AND EVALUATION OF THE
KNIFE FINITE ELEMENT

BRETT W. CLARK
The Development and Evaluation of the
Knife Finite Element

A Thesis
Presented to the
Department of Mechanical Engineering
Brigham Young University

In Partial Fulfillment
of the Requirements for the Degree
Master of Science

by
Brett W. Clark
August 1996
This thesis, by Brett W. Clark, is accepted in its present form by the Department of Mechanical Engineering of Brigham Young University as satisfying the thesis requirements for the degree Master of Science.

Steven E. Benzley, Committee Chair

Jordan J. Cox, Committee Member

Carl D. Sorensen, Committee Member

July 3, 1996
Date

Brent W. Webb, Graduate Coordinator
Table of Contents

Table of Contents .................................................. iii
List of Figures ......................................................... iv
List of Tables ........................................................... v
Acknowledgments ......................................................... vi

1. Objective and Introduction ........................................ 1
   1.1 Objective ..................................................... 1
   1.2 Introduction .................................................. 1
      1.2.1 Plastering ................................................ 3
      1.2.2 Whisker Weaving ......................................... 6
      1.2.3 Other Approaches ....................................... 7
      1.2.4 Problem .................................................. 7
      1.2.5 Proper Development of the Knife Element .......... 8

2. Development of the Knife Finite Element ....................... 9
   2.1 Knife Connectivity and Shape ................................ 9
   2.2 Stiffness Matrix .............................................. 10
      2.2.1 Shape Functions ........................................ 10
      2.2.2 Jacobian ............................................... 13
      2.2.3 Actual Coordinate Derivatives ........................ 14
      2.2.4 B Matrix ............................................... 14
      2.2.5 D Matrix ............................................... 14
      2.2.6 Numerical Integration ................................ 15
   2.3 Alternative Formulation of Stiffness Matrix ............ 15

3. Knife Element Behavior .......................................... 19
   3.1 Approach ..................................................... 19
      3.1.1 Eigenvalue Evaluation ................................. 19
      3.1.2 Tetrahedral Element Comparison ........................ 22
      3.1.3 Patch Test (Axial Loading Evaluation) ............. 25
      3.1.4 Bending Evaluation ................................... 46

4. Conclusions and Usage Requirements ............................ 62
   4.1 Pros and Cons of Solutions to Knife Connectivity ........ 62
      4.1.1 Knife Propagation .................................... 62
      4.1.2 Manually Changing the Knife Connectivity ........... 63
      4.1.3 Filling the Knife Region with Tetrahedral Elements 63
      4.1.4 Leave the Knife Connectivity in the Mesh .......... 63
      4.1.5 Conclusion ............................................ 64
   4.2 Usage Requirements .......................................... 64

References .................................................................. 65
List of Figures

Figure 1.1 - Common occurrence of knife elements. ........................................... 4
Figure 1.2 - Knife shape and connectivity formation. ........................................... 5
Figure 1.3 - Knife propagation. ........................................................................... 6
Figure 2.1 - Derivation of knife element. ............................................................. 9
Figure 2.2 - Node similarities of the knife and hexahedral elements. .................. 11
Figure 2.3 - Two nodes constrained to have the same displacement. ................... 16
Figure 3.1 - Undistorted knife element. ............................................................... 20
Figure 3.2 - Knife shape filled with tetrahedral elements. ................................... 21
Figure 3.3 - Knife shape filled with tetrahedral elements. ................................... 23
Figure 3.4 - Constraints and loading conditions. ............................................... 24
Figure 3.5 - Hexahedral mesh with an embedded knife. ....................................... 26
Figure 3.6 - All-hexahedral mesh. ..................................................................... 27
Figure 3.7 - Node numbering for all-hexahedral mesh. ....................................... 28
Figure 3.8 - Node coordinates for all-hexahedral mesh. ....................................... 28
Figure 3.9 - Loading in the +X direction. ........................................................... 30
Figure 3.10 - Constraints for +X direction loading. .......................................... 31
Figure 3.11 - Node numbering for embedded knife mesh. ................................... 33
Figure 3.12 - Node coordinates for embedded knife mesh. ................................ 34
Figure 3.13 - Loading in the +X direction. ......................................................... 35
Figure 3.14 - Constraints for +X direction loading. ......................................... 36
Figure 3.15 - Loading in the +Y direction. ......................................................... 38
Figure 3.16 - Constraints for +Y direction loading. ......................................... 39
Figure 3.17 - Loading in the +Z direction. ......................................................... 40
Figure 3.18 - Constraints for +Z direction loading. ......................................... 41
Figure 3.19 - Loading in the -Z direction. ......................................................... 42
Figure 3.20 - Constraints for -Z direction loading. ......................................... 43
Figure 3.21 - Other knife shapes: distorted knife base. ...................................... 44
Figure 3.22 - Other knife shapes: distorted knife blade. ...................................... 45
Figure 3.23 - Formation of the 2 knife/8 hex mesh. ............................................ 46
Figure 3.24 - Loading in X direction for bending. ............................................. 47
Figure 3.25 - Constraints for loading in X direction. ......................................... 49
Figure 3.26 - Linear approximation of bending. ............................................... 50
Figure 3.27 - Loading in X direction for bending. ............................................. 51
Figure 3.28 - Mesh configuration for bending evaluation (20 knives.) .................. 54
Figure 3.29 - All-hexahedral beam mesh. ......................................................... 55
Figure 3.30 - Loading condition to simulate a bending moment. ......................... 56
Figure 3.31 - Four and eight knife meshes. ....................................................... 57
Figure 3.32 - Plot of stress ratio vs. percent knife-hex deviation. ....................... 60
List of Tables

Table 3-1 - Knife element eigenvalues. ............................................ 21
Table 3-2 - Eigenvalue comparison (knife vs. tetrahedra.) .................... 22
Table 3-3 - Node displacements (knife vs. tetrahedra.) ........................ 24
Table 3-4 - Nodal displacements in +X direction (all-hex mesh.) .......... 32
Table 3-5 - Nodal displacements in +X direction. ............................... 37
Table 3-6 - Nodal displacements in +Y direction. ............................... 39
Table 3-7 - Nodal displacements in +Z direction. ............................... 41
Table 3-8 - Nodal displacements in -Z direction. ............................... 43
Table 3-9 - Displacements for bending conditions. ............................ 58
Table 3-10 - Nodal displacements for bending (reduced integration.) .... 58
Acknowledgments

I give special thanks to my wife Laura who has supported me completely in all of my school work up through the completion of this thesis. Also, I thank my father for much-needed help in the editing process. I recognize and thank Sandia National Laboratories for providing and funding this research. Finally, I thank my committee chair and members (Steven E. Benzley, Jordan J. Cox, and Carl D. Sorensen) for all of their time and effort in helping me through this research. They were always more than willing to meet with me and give me the help I needed.

Thank you.
Chapter 1

1. Objective and Introduction

1.1 Objective

The objective of this thesis is to show that the knife finite element is a viable element and produces results that are accurate enough to use in finite element analysis.

1.2 Introduction

As the need to accurately analyze and predict physical phenomena in engineering settings has grown, so has the use of finite element analysis. Finite element analysis is a method that provides approximate solutions to the differential equations that govern many of these phenomena such as heat transfer and elasticity. This analysis method approximates the solution by breaking up the continuous domain into discrete regions and solving a system of equations for this domain. The finite element procedure can be divided into three areas: pre-processing, analysis, and post-processing.

Pre-processing consists of generating a mesh to represent the discrete domain. A mesh consists of nodes (coordinate locations) representing discrete locations through the domain. These nodes are connected in an ordered manner to form finite elements. There are many methods for generating meshes. Two types of elements, or node connectivity, are common today for fully three-dimensional analysis. One produces tetrahedral-shaped (4 triangular faced) mesh elements and the other produces hexahedral-shaped (6
quadrilateral faced) mesh elements. Other functions of pre-processing include applying
boundary conditions, specifying loads, and identifying material properties.

After a mesh is generated, a system of algebraic equations that approximate the
governing differential equation posed over the discrete domain, is formulated and solved.
Typical solutions for elasticity problems include displacements, stresses, and strains
produced by the prescribed applied loads and boundary conditions. These equations are
solved using a matrix solver.

After the solution is completed, post-processing is used to visualize the results in
an understandable format. For stress analysis, the different magnitudes of stress could be
represented by different colors, continuously shaded over the entire part. Also, the
displacement could be applied to the body to show the deformed shape.

The three functions of finite element methods defined above, have, to some
extent, been automated by computers. Analysis and post-processing software packages
are widely available. However, an important aspect of automated mesh generation is still
in the stages of research and development. This aspect is the filling of an arbitrarily
shaped volume with all-hexahedral elements. It is relatively simple to fill an arbitrary
volume with tetrahedral elements and many algorithms for doing this have been
developed and are in use in automatic mesh generators. \(^1,2,3\) However, because
comparisons between tetrahedral elements and hexahedral elements show that hexahedral
elements out-perform tetrahedral elements in many areas, \(^4\) robust algorithms for
automatic, all-hexahedral mesh generators are still being developed. \(^5,6\)
Representing the physical object with an appropriate mesh is essential to accurately model physical responses with the finite element method. Just as the shape of the mesh element (tetrahedral or hexahedral) is important, so also is the amount of distortion in the element. Highly distorted mesh elements produce less accurate results. Thus, robust mesh generators are required to create meshes containing optimally-shaped elements. In some cases, where the physical object to be meshed is square and prismatic, it is easy to generate a mesh containing nicely shaped hexahedral elements. However, when curved and tapering surfaces form the boundary of the object, well-formed all-hexahedral meshes are difficult to generate.

An all-hexahedral mesh generator is being developed at Sandia National Laboratories. This package includes advancing front methods called Plastering and Whisker Weaving.

1.2.1 Plastering

The plastering algorithm builds mesh elements based on local geometric considerations. "The plastering algorithm forms a hexahedral mesh by successively adding [hexahedral] elements to the interior of a closed volume" (a surface mesh of quadrilaterals must exist in order to plaster the volume). This approach is attractive for two reasons. First, since the original exterior boundary is used to create elements, the surface mesh quality is preserved. Second, the mesh generated in a volume is independent of the volume's orientation. The plastering process proceeds by building hexes one face at a time. The difficulty in this algorithm is deciding where to project new
nodes when attempting to build a new face. Ideally, as would occur in a perfect cube, faces could be inserted orthogonally to existing faces resulting in a mesh of smaller cubes or hexes. However, when irregularly-shaped volumes are meshed, faces are often inserted at angles that are not orthogonal to existing faces. The result is a mesh with gaps between hexahedral elements and a resulting non-ideal node connectivity. The resulting non-ideal connectivity results in what are called "knife" (or occasionally "wedge") elements. An example of a knife element is shown in Figure 1.1.

Figure 1.1 - Common occurrence of knife elements.

Knife elements are simply degenerate hexahedral elements. The knife shape and connectivity is formed by collapsing two opposing nodes on one face of a hexahedron as shown in Figure 1.2.
Since embedded knife elements are not currently acceptable, they must be removed from the mesh. Thus, when a knife is detected in the mesh, attempts are made to propagate the knife either toward the middle of the current plastering boundary or out toward the surface of the volume by a) collapsing the base, b) opening the blade, or c) intersecting the knife with another knife to form a hexahedron (see Figure 1.3.)
While this procedure often works, propagating the knife to the outer surface of the volume changes the surface mesh which is often the most critical part of the mesh or has to interface with a surface on an adjacent volume. Thus, knife elements may need to be present in a completed mesh.

1.2.2 Whisker Weaving

Whisker weaving is another algorithm for creating an all-hexahedral mesh. Unlike the geometrically-based plastering algorithm, whisker weaving is a connectivity-based algorithm. The mesh connectivity for a given volume is formulated using only the all-quadrilateral surface mesh of the volume. This algorithm is based on a dual representation of mesh entities called the Spatial Twist Continuum. Once the
connectivity of the mesh is formulated, the hexahedral elements are created and the mesh is smoothed. Except for the surface mesh and final smoothing, no geometric considerations are taken when generating the volume mesh. When invalid connectivity is made during the whisker weaving process, knife elements are commonly formed. Similar to plastering, the knife elements must either be removed from the mesh by propagation or left to exist in the mesh.

1.2.3 Other Approaches

Other approaches to all-hexahedral meshes that will not be discussed here include Sweeping, Sub-mapping,\(^1\) and automatic decomposition via medial axis transformations.\(^2\) These approaches do not create knife elements. However, they are somewhat restricted in the geometry they can mesh when transitioning from a large number of elements to a small number of elements and vice versa.

1.2.4 Problem

Knife elements result in attempts to create all-hexahedral meshes. Because of the invalid connectivity (only 7 nodes), the knife element cannot be incorporated as a hexahedron in the analysis software. One solution to this problem is to generate meshes that don’t contain knife elements. This, of course, is the ultimate goal. However, while this is not yet accomplished, other alternatives must be sought. Another solution is to remove the knife element from the mesh using propagation methods. This has proven to work in some cases, but often adversely affects other parts of the mesh. Another solution is to add another node at the same coordinate location as the node in the center of the
knife blade and then adjust the connectivity so as to form a degenerate hexahedron. Like the knife propagation solution, this solution would simply push the bad connectivity to adjacent elements and eventually to the surface mesh if continued. Yet another solution is to fill the region bounded by the knife connectivity with tetrahedral elements. This would require, however, transition elements between the quadrilateral faces of the neighboring hexahedral elements and the triangular faces of the tetrahedral elements. A final solution is to leave the knife elements in the mesh. This thesis proposes that if a finite element with the knife shape and connectivity produced acceptable results during analysis, leaving the knife in the mesh would be an acceptable solution.

1.2.5 Proper Development of the Knife Element

Assuming knife elements are left in the three-dimensional mesh, they must be viable elements. This thesis develops the proper stiffness matrix for an element with the knife shape and connectivity. The behavior of this element is investigated and limitations on its use are presented.
Chapter 2

2. Development of the Knife Finite Element

2.1 Knife Connectivity and Shape

The knife element can be derived from a hexahedral element by collapsing two of the opposing nodes on one face as shown in Figure 2.1.

![Figure 2.1 - Derivation of knife element.](image)

The result is a three-dimensional element made up of five quadrilateral faces.
2.2 **Stiffness Matrix**

The process for generating the stiffness matrix for the knife element is outlined below and follows the procedure suggested in finite element textbooks.\textsuperscript{13,14} It will be discussed in detail in the following sections.

The process to create a finite element stiffness matrix includes the following steps:

- Derive the shape functions of the element with respect to natural coordinates \((\zeta, \eta, \zeta)\).
- Calculate the derivative of the shape functions with respect to natural coordinates.
- Find the Jacobian matrix, its determinant, and inverse.
- Calculate derivatives of shape functions with respect to actual coordinates \((x, y, z)\).
- Calculate B matrix where \([B] = \) product of Boolean matrix and actual coordinate derivatives matrix.
- Calculate D matrix where \([D] = \) the material property matrix.
- Numerically integrate to get the K matrix where \([K] = \) stiffness matrix of element.

### 2.2.1 Shape Functions

*Shape or interpolation functions* are mathematical representations that describe the shape of a finite element. They span the volume of a finite element. These functions of a finite element are formulated so that they interpolate the unknown variables of the element and define the characteristics of the regions between the nodes based on the position with respect to each node. In this way, they can be used as weighting functions for different positions within the finite element. The shape function matrix \([N]\) for the
knife element is derived from the shape functions of a hexahedral element. Figure 2.2 illustrates the similarities between the nodes of a hexahedron and those of a knife.

A top view of the knife is shown to illustrate the warped faces of the knife. The shape functions for nodes 1 through 5 and 7 of the knife element are the same as those for the respective nodes of the hexahedral element. Since the knife element was formed by collapsing nodes 6 and 8 of the hexahedral element (see Figure 2.2), the shape function for node 6 of the knife element is obtained by adding the shape functions of nodes 6 and 8 of the hexahedral element. The shape functions for both the hexahedron and knife are shown below.
Hexahedron

\[ N_1 = \frac{1}{8}(1-\xi)(1-\eta)(1-\zeta) \]
\[ N_2 = \frac{1}{8}(1+\xi)(1-\eta)(1-\zeta) \]
\[ N_3 = \frac{1}{8}(1+\xi)(1+\eta)(1-\zeta) \]
\[ N_4 = \frac{1}{8}(1-\xi)(1+\eta)(1-\zeta) \]
\[ N_5 = \frac{1}{8}(1-\xi)(1-\eta)(1+\zeta) \]
\[ N_6 = \frac{1}{8}(1+\xi)(1-\eta)(1+\zeta) \]
\[ N_7 = \frac{1}{8}(1+\xi)(1+\eta)(1+\zeta) \]
\[ N_8 = \frac{1}{8}(1-\xi)(1+\eta)(1+\zeta) \]

Knife

\[ N_1 = \frac{1}{8}(1-\xi)(1-\eta)(1-\zeta) \]
\[ N_2 = \frac{1}{8}(1+\xi)(1-\eta)(1-\zeta) \]
\[ N_3 = \frac{1}{8}(1+\xi)(1+\eta)(1-\zeta) \]
\[ N_4 = \frac{1}{8}(1-\xi)(1+\eta)(1-\zeta) \]
\[ N_5 = \frac{1}{8}(1-\xi)(1-\eta)(1+\zeta) \]
\[ N_6 = \frac{1}{4}(1-\xi\eta)(1+\zeta) \]
\[ N_7 = \frac{1}{8}(1+\xi)(1+\eta)(1+\zeta) \]
\[ N_8 = \frac{1}{8}(1-\xi)(1+\eta)(1+\zeta) \]

It should be noted that since \( N_6 \) for the knife comes from collapsing nodes 6 and 8 of the hexahedron, \( N_6 \) equals 1 when \( \xi \) and \( \eta \) equal 1 and -1 or -1 and 1 (the coordinates of nodes 6 and 8) respectively. The ability of the knife element shape functions to accurately span the volume of the knife was verified by calculating the volume of the knife using the shape functions and comparing it to the volume calculated using a closed form solution. The volume was calculated using the shape functions by numerically integrating the shape functions using Gaussian integration as will be explained later. The closed form solution for the volume was calculated using classical integration methods. Both methods of calculating the volume resulted in a knife volume of 0.5 units\(^3\). This value came from collapsing a face of a unit hexahedron to get the knife shape.

The next step to generate the stiffness matrix is to calculate the derivatives of the shape functions with respect to each of the natural coordinates (\( \xi \), \( \eta \), and \( \zeta \)). The derivatives of the shape functions for the knife element are shown below.
\[
\begin{align*}
N_{1,\xi} &= -1/8(1-\eta)(1-\zeta) \\
N_{2,\xi} &= 1/8(1-\eta)(1-\zeta) \\
N_{3,\xi} &= 1/8(1+\eta)(1-\zeta) \\
N_{4,\xi} &= -1/8(1+\eta)(1-\zeta) \\
N_{5,\xi} &= -1/8(1-\xi)(1+\zeta) \\
N_{6,\xi} &= -1/4(1+\zeta)(\eta) \\
N_{7,\xi} &= 1/8(1+\eta)(1+\zeta)
\end{align*}
\]

\[
\begin{align*}
N_{1,\eta} &= -1/8(1-\xi)(1-\zeta) \\
N_{2,\eta} &= -1/8(1+\xi)(1-\zeta) \\
N_{3,\eta} &= 1/8(1+\xi)(1-\zeta) \\
N_{4,\eta} &= 1/8(1-\xi)(1-\zeta) \\
N_{5,\eta} &= 1/8(1+\xi)(1+\zeta) \\
N_{6,\eta} &= -1/4(1+\zeta)(\xi) \\
N_{7,\eta} &= 1/8(1-\xi)(1+\zeta)
\end{align*}
\]

\[
\begin{align*}
N_{1,\zeta} &= -1/8(1-\xi)(1-\eta) \\
N_{2,\zeta} &= -1/8(1+\xi)(1-\eta) \\
N_{3,\zeta} &= -1/8(1+\xi)(1+\eta) \\
N_{4,\zeta} &= -1/8(1+\xi)(1+\eta) \\
N_{5,\zeta} &= 1/8(1-\xi)(1-\eta) \\
N_{6,\zeta} &= 1/4(1+\zeta)(\eta) \\
N_{7,\zeta} &= 1/8(1+\xi)(1+\eta)
\end{align*}
\]

### 2.2.2 Jacobian

Next, the Jacobian matrix, its determinant, and inverse are calculated. The entries in the Jacobian matrix are calculated using the shape function derivatives in natural coordinates and the actual coordinates of the nodes of the element as (the notation "\(x,_{\xi}\)"

means the derivative of \(x\) with respect to \(\xi\):

\[
[J] = \begin{bmatrix}
x_{,\xi} & y_{,\xi} & z_{,\xi} \\
x_{,\eta} & y_{,\eta} & z_{,\eta} \\
x_{,\zeta} & y_{,\zeta} & z_{,\zeta}
\end{bmatrix}
\]

(2.1)

\[
x_{,\xi} = N_{1,\xi} x_1 + N_{2,\xi} x_2 + N_{3,\xi} x_3 + \ldots + N_{n,\xi} x_n \\
y_{,\xi} = N_{1,\xi} y_1 + N_{2,\xi} y_2 + N_{3,\xi} y_3 + \ldots + N_{n,\xi} y_n \\
z_{,\xi} = N_{1,\xi} z_1 + N_{2,\xi} z_2 + N_{3,\xi} z_3 + \ldots + N_{n,\xi} z_n
\]

\ldots

\[
y_{,\zeta} = N_{1,\zeta} y_1 + N_{2,\zeta} y_2 + N_{3,\zeta} y_3 + \ldots + N_{n,\zeta} y_n \\
z_{,\zeta} = N_{1,\zeta} z_1 + N_{2,\zeta} z_2 + N_{3,\zeta} z_3 + \ldots + N_{n,\zeta} z_n
\]

(2.2)

After the Jacobian matrix is determined, the determinant and inverse are calculated.
2.2.3 Actual Coordinate Derivatives

The derivatives of the shape functions with respect to actual coordinates are calculated using the Jacobian inverse and the derivatives of the shape functions with respect to natural coordinates using the following equation:

\[
\begin{bmatrix}
\phi_x \\
\phi_y \\
\phi_z
\end{bmatrix}
= \begin{bmatrix}
\phi_x \\
\phi_y \\
\phi_z
\end{bmatrix} \cdot \begin{bmatrix}
\Gamma_x \\
\Gamma_y \\
\Gamma_z
\end{bmatrix}
\text{where } \begin{bmatrix}
\Gamma_x \\
\Gamma_y \\
\Gamma_z
\end{bmatrix} = \left[ J \right]^{-1}
\tag{2.3}
\]

2.2.4 B Matrix

The B matrix is made up of the product of the above calculated actual coordinate derivatives and a Boolean matrix H as shown below:

\[ [B] = [H] \cdot [N_{xyz}] \text{ where } [N_{xyz}] \text{ is the actual coordinate derivative matrix} \]

\[
[B] = \begin{bmatrix}
1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\
0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 & 0 & 1 & 0 & 0 & 0
\end{bmatrix}
\]

and \[ [H] = \begin{bmatrix}
1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\
0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 & 0 & 1 & 0 & 0 & 0
\end{bmatrix} \]

2.2.5 D Matrix

The D Matrix contains the material characteristics and is often specified using Poisson’s ratio \( \nu \), and the modulus of elasticity \( E \) as follows:
Once the \([B]\) matrix, Jacobian, and \([D]\) matrix are calculated, Gaussian Quadrature is used to determine the stiffness matrix, \([K]\) as:

\[
[K] = \int_{-1}^{1} \int_{-1}^{1} \int_{-1}^{1} [B]^T [D] B \, J d\xi \, d\eta \, d\zeta
\]

where \(J\) is the Jacobian determinant. For a hexahedron stiffness matrix, 2x2x2 Gaussian Quadrature is sufficient for convergence.15 The order of integration for the knife element is an area of research that can be done but will not be included in this thesis. For the knife element, 4x4x4 Gaussian Quadrature was chosen arbitrarily.

### 2.3 Alternative Formulation of Stiffness Matrix

There exists an alternative method for formulating the knife element stiffness matrix that produces exactly the same stiffness matrix as the process described above. The above formulation was based on a seven-node element. The stiffness matrix for the knife element can also be formulated by degenerating an eight-node element and constraining two of the nodes to have the same physical location as well as to have the same displacement. This is illustrated in Figure 2.3.
In order to modify the stiffness matrix of the hexahedral element to obtain that of the knife element, we start with the matrix equation $[K][u] = [F]$ as shown below (only one degree of freedom per node will be used for illustration purposes.) Nodes 6 and 8 are collapsed to the same location as depicted in Figure 2.3.

$$
\begin{bmatrix}
    k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} & k_{17} & k_{18} \\
    k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} & k_{27} & k_{28} \\
    k_{31} & k_{32} & k_{33} & k_{34} & k_{35} & k_{36} & k_{37} & k_{38} \\
    k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} & k_{47} & k_{48} \\
    k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} & k_{57} & k_{58} \\
    k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} & k_{67} & k_{68} \\
    k_{71} & k_{72} & k_{73} & k_{74} & k_{75} & k_{76} & k_{77} & k_{78} \\
    k_{81} & k_{82} & k_{83} & k_{84} & k_{85} & k_{86} & k_{87} & k_{88}
\end{bmatrix}
\begin{bmatrix}
    u_1 \\
    u_2 \\
    u_3 \\
    u_4 \\
    u_5 \\
    u_6 \\
    u_7 \\
    u_8
\end{bmatrix} = 
\begin{bmatrix}
    f_1 \\
    f_2 \\
    f_3 \\
    f_4 \\
    f_5 \\
    f_6 \\
    f_7 \\
    f_8
\end{bmatrix}
$$
The objective is to constrain the displacements $u_6$ and $u_8$ (the nodes on the knife blade that are collapsed onto each other) so that they have the same value. This can be accomplished by introducing the linear transformation $[u] = [L][q]$ where $[u]$ is the displacement vector shown above, $[q]$ is a displacement vector with only seven elements representing the displacements of the seven nodes of the knife element, and $[L]$ is an $8 \times 7$ transformation matrix as shown below.

$$
\begin{bmatrix}
1 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 1 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 1 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 1 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 1 \\
0 & 0 & 0 & 0 & 0 & 0 & 1
\end{bmatrix}
$$

The transformation matrix $[L]$ constrains displacements $u_6$ and $u_8$ in the $u$ displacement vector to be the same as $q_6$ in the new $[q]$ displacement vector. The initial equation $[K][u] = [F]$ now becomes $[K][L][q] = [F]$ by substituting $[L][q]$ in for $[u]$. By pre-multiplying the whole equation by $[L]^T$, we get $([L]^T[K][L])[q] = ([L]^T[F])$ where $([L]^T[K][L])$ is now the stiffness matrix for the seven-noded knife element, $[q]$ is the displacement vector for the knife element, and $([L]^T[F])$ is the force vector for the knife element. These relationships can be quite useful for dealing with knife connectivity using analysis code that doesn’t support the knife finite element. If the knife connectivity arises, a node can be introduced on the knife blade and constrained to have the same
displacement as the node that already exists in that position by using the above matrix relationships. It should be noted that the new node should not be connected to any elements but the knife element because doing this would just propagate the knife connectivity to a new location. This solution produces identical results to using the knife finite element if the analysis code does not support the knife finite element. However, extra work is involved.
Chapter 3

3. Knife Element Behavior

3.1 Approach

The main objective, as stated before, in evaluating the properties of the knife element is to determine if it produces satisfactory results when used in a finite element model. The evaluation approach described below is used to predict, both by theory and practice, how the knife element performs. The following four methods are used to evaluate the accuracy of the knife element: 1) eigenvalue evaluation, 2) tetrahedral element comparison, 3) patch test (axial loading evaluation), and 4) bending evaluation.

3.1.1 Eigenvalue Evaluation

The first part of the evaluation approach focuses on analyzing the characteristics of the knife element stiffness matrix. The viability of a finite element can partially be determined by examining the eigenvalues extracted from the element stiffness matrix. Each eigenvalue extracted from the stiffness matrix is proportional to the strain energy associated with a corresponding deformation mode (i.e. eigenvector.) Zero rigid body strain energy means that the element will have no strain energy induced when it displaces under solely rigid body motion. This is a necessary condition for convergence. The number of rigid body modes for a three-dimensional solid finite element is six (i.e. three translations in the three coordinate directions and three rotations about the three coordinate axes.) Thus, six zero-valued eigenvalues are required for the knife element.
Although the zero rigid body strain energy does not guarantee good performance, it provides a theoretical check for the knife element viability.

A standard, undistorted knife element (see Figure 3.1) was constructed and evaluated.

Figure 3.1 - Undistorted knife element.

The knife was formed by collapsing two nodes of a unit cube as described earlier. The stiffness matrix was calculated, and the eigenvalues extracted. As expected, six of the twenty-one (i.e. 7 nodes x 3 degrees of freedom per node) stiffness matrix eigenvalues had a value of zero representing the six rigid body modes. A full list of the eigenvalues for an undeformed knife element is given below in Table 3-1.
Table 3-1 - Knife element eigenvalues.

<table>
<thead>
<tr>
<th>Knife Eigenvalues (x 1.00E7)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000</td>
</tr>
<tr>
<td>0.0000</td>
</tr>
<tr>
<td>0.0000</td>
</tr>
<tr>
<td>0.0000</td>
</tr>
<tr>
<td>0.0000</td>
</tr>
<tr>
<td>0.0000</td>
</tr>
<tr>
<td>0.1534</td>
</tr>
</tbody>
</table>

For comparison, the eigenvalues for a configuration of tetrahedral elements that closely resembles the knife shape were calculated. The tetrahedral configuration is shown below in Figure 3.2.

Figure 3.2 - Knife shape filled with tetrahedral elements.

The eigenvalues for the tetrahedral configuration are shown in Table 3-2 along with those from the knife element.
Table 3-2 - Eigenvalue comparison (knife vs. tetrahedra.)

<table>
<thead>
<tr>
<th></th>
<th>Knife</th>
<th>6 Tet Mesh</th>
<th>Knife</th>
<th>6 Tet Mesh</th>
<th>Knife</th>
<th>6 Tet Mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000</td>
<td>0.0000</td>
<td>0.1860</td>
<td>0.8252</td>
<td>0.8881</td>
<td>2.0675</td>
<td></td>
</tr>
<tr>
<td>0.0000</td>
<td>0.0000</td>
<td>0.2852</td>
<td>0.8782</td>
<td>0.9043</td>
<td>3.1734</td>
<td></td>
</tr>
<tr>
<td>0.0000</td>
<td>0.0000</td>
<td>0.3515</td>
<td>0.8859</td>
<td>0.9113</td>
<td>3.4615</td>
<td></td>
</tr>
<tr>
<td>0.0000</td>
<td>0.0000</td>
<td>0.4769</td>
<td>1.1538</td>
<td>2.3071</td>
<td>3.5898</td>
<td></td>
</tr>
<tr>
<td>0.0000</td>
<td>0.0000</td>
<td>0.6168</td>
<td>1.2000</td>
<td>2.3132</td>
<td>5.2922</td>
<td></td>
</tr>
<tr>
<td>0.0000</td>
<td>0.0000</td>
<td>0.6538</td>
<td>1.2587</td>
<td>3.7882</td>
<td>6.5226</td>
<td></td>
</tr>
<tr>
<td>0.1534</td>
<td>0.7224</td>
<td>0.6924</td>
<td>1.9787</td>
<td>6.0616</td>
<td>8.2400</td>
<td></td>
</tr>
</tbody>
</table>

The eigenvalues for the tetrahedral element mesh configuration are larger than the single knife, indicating higher deformation mode strain energy and a stiffer element configuration.

Summary of Eigenvalue Evaluation

The eigenvalues of the knife element were extracted from the stiffness matrix. Six rigid body modes were present. The knife element eigenvalues were compared with the eigenvalues extracted from a tetrahedral element stiffness matrix of the same shape and the eigenvalues of the knife element were lower indicating a less stiff element.

3.1.2 Tetrahedral Element Comparison

The rest of the evaluation approach is aimed at analyzing the performance of the knife element when it is actually used in a mesh. This section compares the knife element to the configuration of tetrahedral elements used in the last section and shown in Figure 3.3.
One of the mentioned solutions to the problem of knife shape and connectivity in a mesh was to fill the region bounded by the knife nodes with tetrahedral elements. To determine if this is a viable solution, a test to compare the stiffness of the knife element and the tetrahedral elements was performed. In the previous section, an eigenvalue evaluation showed that the knife element is indeed less stiff than the tetrahedral elements. Here, loads will be applied to both the knife element and the tetrahedral elements to see which one displaces more.

Figure 3.4 shows the constraints and loading conditions that were applied to the two meshes.
Figure 3.4 - Constraints and loading conditions.

The displacements in the z direction at nodes A, B, and C for the two different meshes are shown in Table 3-3.

Table 3-3 - Node displacements (knife vs. tetrahedra.)

<table>
<thead>
<tr>
<th>Node</th>
<th>Displacements (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Knife</td>
<td>A</td>
</tr>
<tr>
<td></td>
<td>5.805</td>
</tr>
<tr>
<td>6 Tet Mesh</td>
<td>4.000</td>
</tr>
</tbody>
</table>

From the displacements, it is clear that the knife is less stiff than a knife shape filled with tetrahedral elements.

Summary of the Tetrahedral Element Comparison

In order to compare the stiffness of the knife element with that of tetrahedral elements, a knife-like shape was filled with six tetrahedral elements. Equivalent loads were applied to both the knife element and the tetrahedral mesh in order to see which one
would displace more. The knife element displaced more than the tetrahedral elements indicating that it is less stiff. Consequently, a knife element is preferrable to a knife shape filled with tetrahedral elements.

3.1.3 Patch Test (Axial Loading Evaluation)

Next, the performance of the knife under axial loading was determined using the *patch test*. The patch test is a commonly used method for verifying the validity of finite elements by analyzing the deflections and stresses in patches of elements due to applied loads or displacements. This is accomplished by embedding one or more knife elements into an all-hexahedral mesh, applying a force to the mesh, and then calculating the resulting deflection. Results are then compared to the results obtained using a mesh without embedded knife elements. Thus, the effects of the embedded knife element(s) on the finite element analysis can be observed. In the axial loading cases, the stresses throughout each element of the mesh were also calculated. These stresses were compared to the applied stress to insure that the stress field throughout the mesh was consistent with the applied stress. These deflection and stress analyses are variations of the patch test.

One Knife in Four Hexes

The first mesh analyzed is shown in Figure 3.5.
The knife blade runs along the bottom surface of the mesh and the base lies in the top surface of the mesh. In this way, any knife-hex deviation was easily detected since the forces were applied to the surface of the mesh and the displacements were also measured on the surface of the mesh.

For comparison purposes and to confirm that the software developed for this thesis was performing correctly, a mesh with the same surface size and shape was constructed and analyzed using only hexahedral elements as shown in Figure 3.6. The displacement analysis was first done on this mesh to determine how the nodes in the mesh should displace due to an applied load.
The all-hexahedral mesh was constructed so that it was exactly the same size as the mesh containing the embedded knife. Figure 3.7 and Figure 3.8 illustrate the numbering and coordinates of the nodes in the all-hexahedral mesh.
Figure 3.7 - Node numbering for all-hexahedral mesh.

<table>
<thead>
<tr>
<th>Node</th>
<th>x</th>
<th>y</th>
<th>z</th>
<th>Node</th>
<th>x</th>
<th>y</th>
<th>z</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>9</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>10</td>
<td>1</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>11</td>
<td>2</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>12</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>13</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>1</td>
<td>0</td>
<td>14</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>6</td>
<td>0</td>
<td>2</td>
<td>0</td>
<td>15</td>
<td>0</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>2</td>
<td>0</td>
<td>16</td>
<td>1</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>8</td>
<td>2</td>
<td>2</td>
<td>0</td>
<td>17</td>
<td>2</td>
<td>2</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 3.8 - Node coordinates for all-hexahedral mesh.
To compute the displacements due to applied loads on the mesh, the stiffness matrix was formulated and displacements calculated as:

$$[u] = [K]^{-1} \bullet [F]$$

where $[u]$ is the displacement vector, $[K]^{-1}$ is the inverse of the mesh system stiffness matrix, and $[F]$ is the loading vector. As usual, the element stiffness matrices were calculated for each element in the mesh and the system stiffness matrix was assembled using the components from the element matrices.

The loading vector for finite element analysis is formed as:

$$[F] = \int_V [B]^T [E] \{\varepsilon_0\} dV - \int_V [B]^T \{\sigma_0\} dV$$
$$+ \int_V [N]^T \{F\} dV + \int_S [N]^T \{\Phi\} dS$$

where the different terms in the loading vector come from body forces, surface tractions, etc. As in the case of the stiffness matrix formation, Gaussian integration is used to numerically evaluate the above integrals. The loading cases that follow only require the traction term in the above loading vector equation. Figure 3.9 shows the loading case that was applied in the positive x direction. Loading cases for the y and z directions were also performed in separate tests.
In order to prevent rigid body motion of the mesh, constraints were applied as shown in Figure 3.10. The arrows represent the degrees of freedom that are constrained at the respective nodes.
Figure 3.10 - Constraints for +X direction loading.

The values of $E$ (the modulus of elasticity) and $v$ (Poisson’s ratio) for all of the evaluations performed in this study were 30,000,000 psi and 0.3 respectively. Table 3-4 shows the displacements that resulted at each node from the loading case in the positive $x$ direction.
Table 3-4 - Nodal displacements in +X direction (all-hex mesh.)

<table>
<thead>
<tr>
<th>DOF</th>
<th>Disp. (ln.)</th>
<th>DOF</th>
<th>Disp. (ln.)</th>
<th>DOF</th>
<th>Disp. (ln.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1x</td>
<td>1.000</td>
<td>6z</td>
<td>0.000</td>
<td>12z</td>
<td>-0.300</td>
</tr>
<tr>
<td>1y</td>
<td>0.000</td>
<td>7x</td>
<td>1.000</td>
<td>13x</td>
<td>1.000</td>
</tr>
<tr>
<td>1z</td>
<td>0.000</td>
<td>7y</td>
<td>-0.600</td>
<td>13y</td>
<td>-0.300</td>
</tr>
<tr>
<td>2x</td>
<td>2.000</td>
<td>7z</td>
<td>0.000</td>
<td>13z</td>
<td>-0.300</td>
</tr>
<tr>
<td>2y</td>
<td>0.000</td>
<td>8x</td>
<td>2.000</td>
<td>14x</td>
<td>2.000</td>
</tr>
<tr>
<td>2z</td>
<td>0.000</td>
<td>8y</td>
<td>-0.600</td>
<td>14y</td>
<td>-0.300</td>
</tr>
<tr>
<td>3y</td>
<td>-0.300</td>
<td>8z</td>
<td>0.000</td>
<td>14z</td>
<td>-0.300</td>
</tr>
<tr>
<td>3z</td>
<td>0.000</td>
<td>9z</td>
<td>-0.300</td>
<td>15y</td>
<td>-0.600</td>
</tr>
<tr>
<td>4x</td>
<td>1.000</td>
<td>10x</td>
<td>1.000</td>
<td>15z</td>
<td>-0.300</td>
</tr>
<tr>
<td>4y</td>
<td>-0.300</td>
<td>10y</td>
<td>0.000</td>
<td>16x</td>
<td>1.000</td>
</tr>
<tr>
<td>4z</td>
<td>0.000</td>
<td>10z</td>
<td>-0.300</td>
<td>16y</td>
<td>-0.600</td>
</tr>
<tr>
<td>5x</td>
<td>2.000</td>
<td>11x</td>
<td>2.000</td>
<td>16z</td>
<td>-0.300</td>
</tr>
<tr>
<td>5y</td>
<td>-0.300</td>
<td>11y</td>
<td>0.000</td>
<td>17x</td>
<td>2.000</td>
</tr>
<tr>
<td>5z</td>
<td>0.000</td>
<td>11z</td>
<td>-0.300</td>
<td>17y</td>
<td>-0.600</td>
</tr>
<tr>
<td>6y</td>
<td>-0.600</td>
<td>12y</td>
<td>-0.300</td>
<td>17z</td>
<td>-0.300</td>
</tr>
</tbody>
</table>

The computed displacements of 1.000 and 2.000 inches in the direction of the applied force correspond exactly to the closed form solution \( \delta = \frac{PL}{AE} \)

where \( \delta \) is the displacement, \( P \) is the total load applied to the mesh face, \( L \) is the length of the mesh in the applied load direction, \( A \) is the cross-sectional area of the mesh face, and \( E \) is the modulus of elasticity. The loading vector was developed so that the applied load per element face would equal the modulus of elasticity \( (30,000,000) \), thus, resulting in a unit displacement per element. Also, the displacements due to Poisson's effect in the directions perpendicular to the load should be 0.3 inches per element. This method was used throughout all of the axial loading cases to determine the loads to apply to each node. This simple, all-hexahedral test case was done for three reasons. First, this case shows what displacements to expect when analyzing the mesh with the embedded knife.
Second, it illustrates the process that was used to apply loads to meshes and measure the resulting displacements. Third, it verifies that the finite element algorithm developed for this thesis is performing correctly.

Next, similar load/displacement analyses were done for the mesh containing four hexahedral elements and one knife element. The node numbering and node coordinates for this mesh are shown in Figure 3.11 and Figure 3.12.

![Figure 3.11 - Node numbering for embedded knife mesh.](image-url)
Figure 3.12 - Node coordinates for embedded knife mesh.

As with the all-hexahedral mesh, the stiffness matrix for the embedded knife mesh was formulated. Again, 2x2x2 Gaussian Quadrature was used to compute the element stiffness matrices for the hexahedral elements. The components of the stiffness matrix for the knife element are not integrated exactly using 2x2x2 Gaussian Quadrature. In fact, integration is not exact even after using a 6x6x6 Gaussian Quadrature. This was an interesting phenomenon due probably to the amount of skew present in the element shape. However, it did not seem to affect the performance of the knife element, especially under pure, axial loading. Therefore, 2x2x2 Gaussian Quadrature was used to compute the knife element stiffness matrix. Loads were applied in the following four directions: 1) positive x direction, 2) positive y direction, 3) positive z direction, and 4) negative z direction. The negative z direction loading case was done because loading
cases in the z directions are not symmetric as are the x and y directions. The following sections show the setup and results for these four loading cases on the embedded knife mesh.

Case 1: +X Direction

The first case was loading in the positive x direction as seen in Figure 3.13.

The constraints required to prevent rigid body motion of the mesh are shown in Figure 3.14.
The displacements for loading in the positive x direction are shown below in Table 3-5. The closed form solution (CFS) for the displacements is also shown for each degree of freedom.
Table 3-5 - Nodal displacements in +X direction.

<table>
<thead>
<tr>
<th>DOF</th>
<th>Disp. (in.)</th>
<th>CFS</th>
<th>DOF</th>
<th>Disp. (in.)</th>
<th>CFS</th>
<th>DOF</th>
<th>Disp. (in.)</th>
<th>CFS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1x</td>
<td>1.000</td>
<td>1.000</td>
<td>7x</td>
<td>1.000</td>
<td>1.000</td>
<td>13y</td>
<td>-0.150</td>
<td>-0.150</td>
</tr>
<tr>
<td>1y</td>
<td>0.000</td>
<td>0.000</td>
<td>7y</td>
<td>-0.600</td>
<td>-0.600</td>
<td>13z</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>1z</td>
<td>0.000</td>
<td>0.000</td>
<td>7z</td>
<td>0.000</td>
<td>0.000</td>
<td>14x</td>
<td>2.000</td>
<td>2.000</td>
</tr>
<tr>
<td>2x</td>
<td>2.000</td>
<td>2.000</td>
<td>8x</td>
<td>2.000</td>
<td>2.000</td>
<td>14y</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>2y</td>
<td>0.000</td>
<td>0.000</td>
<td>8y</td>
<td>-0.600</td>
<td>-0.600</td>
<td>14z</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>2z</td>
<td>0.000</td>
<td>0.000</td>
<td>8z</td>
<td>0.000</td>
<td>0.000</td>
<td>15x</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>3y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>9z</td>
<td>-0.300</td>
<td>-0.300</td>
<td>15y</td>
<td>-0.450</td>
<td>-0.450</td>
</tr>
<tr>
<td>3z</td>
<td>0.000</td>
<td>0.000</td>
<td>10x</td>
<td>1.000</td>
<td>1.000</td>
<td>15z</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>4x</td>
<td>1.000</td>
<td>1.000</td>
<td>10y</td>
<td>0.000</td>
<td>0.000</td>
<td>16x</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>4y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>10z</td>
<td>-0.300</td>
<td>-0.300</td>
<td>16y</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>4z</td>
<td>0.000</td>
<td>0.000</td>
<td>11x</td>
<td>2.000</td>
<td>2.000</td>
<td>17x</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>5x</td>
<td>2.000</td>
<td>2.000</td>
<td>11y</td>
<td>0.000</td>
<td>0.000</td>
<td>17y</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>5y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>11z</td>
<td>-0.300</td>
<td>-0.300</td>
<td>17z</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>5z</td>
<td>0.000</td>
<td>0.000</td>
<td>12y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>18x</td>
<td>2.000</td>
<td>2.000</td>
</tr>
<tr>
<td>6y</td>
<td>-0.600</td>
<td>-0.600</td>
<td>12z</td>
<td>-0.300</td>
<td>-0.300</td>
<td>18y</td>
<td>-0.500</td>
<td>-0.600</td>
</tr>
<tr>
<td>6z</td>
<td>0.000</td>
<td>0.000</td>
<td>13x</td>
<td>1.000</td>
<td>1.000</td>
<td>18z</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
</tbody>
</table>

These nodal displacements are identical to those of the all-hexahedral mesh except in the areas where nodes have different x, y, z coordinates (due to the embedded knife). The displacements obtained by the finite element method also match the closed form displacements exactly.

**Case 2: +Y Direction**

The second loading case was in the positive y direction as seen in Figure 3.15.
Figure 3.15 - Loading in the +Y direction.

The constraints required to prevent rigid body motion of the mesh are shown in Figure 3.16.
The displacements for loading in the positive $y$ direction are shown below in Table 3-6.

<table>
<thead>
<tr>
<th>DOF</th>
<th>Disp. (In.)</th>
<th>CFS</th>
<th>DOF</th>
<th>Disp. (In.)</th>
<th>CFS</th>
<th>DOF</th>
<th>Disp. (In.)</th>
<th>CFS</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0x$</td>
<td>0.600</td>
<td>0.600</td>
<td>$7x$</td>
<td>0.300</td>
<td>0.300</td>
<td>$13z$</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>$0z$</td>
<td>0.000</td>
<td>0.000</td>
<td>$7y$</td>
<td>2.000</td>
<td>2.000</td>
<td>$14x$</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>$1x$</td>
<td>0.300</td>
<td>0.300</td>
<td>$7z$</td>
<td>0.000</td>
<td>0.000</td>
<td>$14y$</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>$1z$</td>
<td>0.000</td>
<td>0.000</td>
<td>$8x$</td>
<td>0.000</td>
<td>0.000</td>
<td>$14z$</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>$3x$</td>
<td>0.600</td>
<td>0.600</td>
<td>$8y$</td>
<td>2.000</td>
<td>2.000</td>
<td>$15x$</td>
<td>0.300</td>
<td>0.300</td>
</tr>
<tr>
<td>$3y$</td>
<td>1.000</td>
<td>1.000</td>
<td>$8z$</td>
<td>0.000</td>
<td>0.000</td>
<td>$15y$</td>
<td>1.500</td>
<td>1.500</td>
</tr>
<tr>
<td>$3z$</td>
<td>0.000</td>
<td>0.000</td>
<td>$9x$</td>
<td>0.600</td>
<td>0.600</td>
<td>$15z$</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>$4x$</td>
<td>0.300</td>
<td>0.300</td>
<td>$9z$</td>
<td>-0.300</td>
<td>-0.300</td>
<td>$16x$</td>
<td>0.600</td>
<td>0.600</td>
</tr>
<tr>
<td>$4y$</td>
<td>1.000</td>
<td>1.000</td>
<td>$10x$</td>
<td>0.300</td>
<td>0.300</td>
<td>$16y$</td>
<td>2.000</td>
<td>2.000</td>
</tr>
<tr>
<td>$4z$</td>
<td>0.000</td>
<td>0.000</td>
<td>$10z$</td>
<td>-0.300</td>
<td>-0.300</td>
<td>$16z$</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>$5x$</td>
<td>0.000</td>
<td>0.000</td>
<td>$11z$</td>
<td>-0.300</td>
<td>-0.300</td>
<td>$17x$</td>
<td>0.300</td>
<td>0.300</td>
</tr>
<tr>
<td>$5y$</td>
<td>1.000</td>
<td>1.000</td>
<td>$12x$</td>
<td>0.600</td>
<td>0.600</td>
<td>$17y$</td>
<td>2.000</td>
<td>2.000</td>
</tr>
<tr>
<td>$5z$</td>
<td>0.000</td>
<td>0.000</td>
<td>$12y$</td>
<td>1.000</td>
<td>1.000</td>
<td>$17z$</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>$6x$</td>
<td>0.600</td>
<td>0.600</td>
<td>$12z$</td>
<td>-0.300</td>
<td>-0.300</td>
<td>$18x$</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>$6y$</td>
<td>2.000</td>
<td>2.000</td>
<td>$13x$</td>
<td>0.300</td>
<td>0.300</td>
<td>$18y$</td>
<td>2.000</td>
<td>2.000</td>
</tr>
<tr>
<td>$6z$</td>
<td>0.000</td>
<td>0.000</td>
<td>$13y$</td>
<td>0.500</td>
<td>0.500</td>
<td>$18z$</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
</tbody>
</table>
Again, the displacements are exactly what would be expected for the given loading case even in the area of the embedded knife.

Case 3: +Z Direction

The third loading case was in the positive z direction as shown in Figure 3.17.

![Figure 3.17 - Loading in the +Z direction.](image)

The constraints required to prevent rigid body motion of the mesh are shown in Figure 3.18.
Figure 3.18 - Constraints for +Z direction loading.

The displacements for loading in the positive z direction are shown below in Table 3-7.

Table 3-7 - Nodal displacements in +Z direction.

<table>
<thead>
<tr>
<th>DOF</th>
<th>Disp. (ln.)</th>
<th>CFS</th>
<th>DOF</th>
<th>Disp. (ln.)</th>
<th>CFS</th>
<th>DOF</th>
<th>Disp. (ln.)</th>
<th>CFS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1x</td>
<td>-0.300</td>
<td>-0.300</td>
<td>9x</td>
<td>0.000</td>
<td>0.000</td>
<td>14x</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>1y</td>
<td>0.000</td>
<td>0.000</td>
<td>9y</td>
<td>0.000</td>
<td>0.000</td>
<td>14y</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>2x</td>
<td>-0.600</td>
<td>-0.600</td>
<td>10z</td>
<td>1.000</td>
<td>1.000</td>
<td>15z</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>2y</td>
<td>0.000</td>
<td>0.000</td>
<td>10y</td>
<td>0.000</td>
<td>0.000</td>
<td>15y</td>
<td>-0.450</td>
<td>-0.450</td>
</tr>
<tr>
<td>3x</td>
<td>0.000</td>
<td>0.000</td>
<td>10z</td>
<td>1.000</td>
<td>1.000</td>
<td>15z</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>3y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>11x</td>
<td>-0.600</td>
<td>-0.600</td>
<td>16x</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>4x</td>
<td>-0.300</td>
<td>-0.300</td>
<td>11y</td>
<td>0.000</td>
<td>0.000</td>
<td>16y</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>4y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>11z</td>
<td>1.000</td>
<td>1.000</td>
<td>16z</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>5x</td>
<td>-0.600</td>
<td>-0.600</td>
<td>12x</td>
<td>0.000</td>
<td>0.000</td>
<td>17x</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>5y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>12y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>17y</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>6x</td>
<td>-0.600</td>
<td>-0.600</td>
<td>12z</td>
<td>1.000</td>
<td>1.000</td>
<td>17z</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>6y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>13x</td>
<td>-0.300</td>
<td>-0.300</td>
<td>18x</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>7x</td>
<td>-0.600</td>
<td>-0.600</td>
<td>13y</td>
<td>-0.150</td>
<td>-0.150</td>
<td>18y</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>7y</td>
<td>-0.600</td>
<td>-0.600</td>
<td>13z</td>
<td>1.000</td>
<td>1.000</td>
<td>18z</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>8x</td>
<td>-0.600</td>
<td>-0.600</td>
<td>13y</td>
<td>-0.150</td>
<td>-0.150</td>
<td>18y</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>8y</td>
<td>-0.600</td>
<td>-0.600</td>
<td>13z</td>
<td>1.000</td>
<td>1.000</td>
<td>18z</td>
<td>1.000</td>
<td>1.000</td>
</tr>
</tbody>
</table>
Case 4: -Z Direction

The final loading case was in the negative z direction as shown in Figure 3.19.

Figure 3.19 - Loading in the -Z direction.

The constraints required to prevent rigid body motion of the mesh are shown in Figure 3.20.
The displacements for loading in the negative z direction are shown below in Table 3-8.

### Table 3-8 - Nodal displacements in -Z direction.

<table>
<thead>
<tr>
<th>DOF</th>
<th>Disp. (ln.)</th>
<th>CFS</th>
<th>DOF</th>
<th>Disp. (ln.)</th>
<th>CFS</th>
<th>DOF</th>
<th>Disp. (ln.)</th>
<th>CFS</th>
</tr>
</thead>
<tbody>
<tr>
<td>0x</td>
<td>0.000</td>
<td>0.000</td>
<td>5x</td>
<td>-0.600</td>
<td>-0.600</td>
<td>11y</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>0y</td>
<td>0.000</td>
<td>0.000</td>
<td>5y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>12x</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>0z</td>
<td>-1.000</td>
<td>-1.000</td>
<td>6x</td>
<td>0.000</td>
<td>0.000</td>
<td>13x</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>1x</td>
<td>-0.300</td>
<td>-0.300</td>
<td>6y</td>
<td>-0.600</td>
<td>-0.600</td>
<td>14x</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>1y</td>
<td>0.000</td>
<td>0.000</td>
<td>6z</td>
<td>-1.000</td>
<td>-1.000</td>
<td>15x</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>1z</td>
<td>-1.000</td>
<td>-1.000</td>
<td>7x</td>
<td>-0.300</td>
<td>-0.300</td>
<td>15y</td>
<td>-0.300</td>
<td>-0.300</td>
</tr>
<tr>
<td>2x</td>
<td>-0.600</td>
<td>-0.600</td>
<td>7y</td>
<td>-0.600</td>
<td>-0.600</td>
<td>16x</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>2y</td>
<td>0.000</td>
<td>0.000</td>
<td>7z</td>
<td>-1.000</td>
<td>-1.000</td>
<td>17x</td>
<td>-0.450</td>
<td>-0.450</td>
</tr>
<tr>
<td>2z</td>
<td>-1.000</td>
<td>-1.000</td>
<td>8x</td>
<td>-0.600</td>
<td>-0.600</td>
<td>18x</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>3x</td>
<td>-0.300</td>
<td>-0.300</td>
<td>8y</td>
<td>-0.600</td>
<td>-0.600</td>
<td>19y</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>3y</td>
<td>-1.000</td>
<td>-1.000</td>
<td>8z</td>
<td>-1.000</td>
<td>-1.000</td>
<td>20y</td>
<td>-0.600</td>
<td>-0.600</td>
</tr>
<tr>
<td>3z</td>
<td>-0.300</td>
<td>-0.300</td>
<td>10x</td>
<td>-0.300</td>
<td>-0.300</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4x</td>
<td>-0.300</td>
<td>-0.300</td>
<td>10y</td>
<td>0.000</td>
<td>0.000</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4y</td>
<td>-0.300</td>
<td>-0.300</td>
<td>11x</td>
<td>-0.600</td>
<td>-0.600</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Other Axial Loading Evaluations

The previous evaluations were all done using an "undistorted" knife element. To evaluate how a distorted knife element behaves, further evaluations are required. The following cases were done using axial loading to investigate distorted knives.

Figure 3.21 shows different knife shapes where the base of the knife was distorted (top view).

Figure 3.21 - Other knife shapes: distorted knife base.

The four different axial loading cases were applied to the five mesh configurations in Figure 3.21 and there was no knife-hex deviation in the resulting nodal displacements.

Also, the blade of the knife was distorted as shown in Figure 3.22 and the same loading cases were applied to each of the three configurations. Again, there was no knife-hex deviation in the resulting nodal displacements.
Summary of Axial Loading

In order to observe the knife element's performance when subjected to axial loads, the knife was embedded in a mesh consisting of four hexahedral elements as shown in Figure 3.5. Axial loads were applied in the positive x, positive y, positive z, and negative z directions. The resulting nodal displacements were then calculated. Clearly, from the results observed, there is no knife-hex deviation in these loading cases. Also, the knife element nodes can be displaced so that the knife element is severely distorted, and there is still no knife-hex deviation under axial loading. The axial loading cases done were all applications of the patch test. Since axial loading creates a linear displacement field within the mesh, passing the patch test insures convergence of this element.\textsuperscript{15}
3.1.4 Bending Evaluation

Not all applied loads are axial. Therefore, this section focuses on the performance of the knife element under bending conditions.

Localized Evaluation (Two Knife Elements in Eight Hex Elements)

This first section will examine the effects of the knife elements on a very small mesh under bending conditions. In order to do this, a new mesh of elements was formed. Figure 3.23 illustrates the new configuration consisting of two knife elements and eight hexahedral elements. This mesh was created by simply joining two meshes derived from the mesh (the original mesh and an inverted mesh) used in the axial loading cases. The new mesh contains two knives in the middle with their bases back-to-back and the knife blades running along the top and bottom of the mesh.

![Figure 3.23 - Formation of the 2 knife/8 hex mesh.](image-url)
Again, unit hexahedral elements were used so that the overall dimensions of the mesh are 2x2x2. As in the axial loading cases, an all-hexahedral mesh was also constructed for comparison purposes. The all-hexahedral mesh was made up of ten hexahedral elements configured so as to represent the knife mesh with hexes oriented similarly to the two knife elements. A bending situation was simulated by applying loads to this all-hexahedral mesh as shown in Figure 3.24.

![Figure 3.24 - Loading in X direction for bending.](image)

The moment couple resulting from the applied loads was calculated using the following formula:

\[ M = (P \times d) \]

where \( P \) is the applied load on one side of the couple (80,000 lbs) and \( d \) is the distance between the two sides of the couple (2 inches.) Displacements for all of the nodes were
calculated. However, only node “A” (circled) will be used to compare displacement results for the various bending cases that follow. This node was chosen because it is the area in the mesh where the largest displacements occur.

A simple beam closed-form solution for the displacement at node A is obtained using the following formula: \[\phi = \frac{ML}{EI}\] (3.1)

where displacement is the rotation about the neutral axis. For small angles, the axial displacement is \(\phi(z)\). In this problem, \(z = 1.0\), thus, \(\phi = \delta\). This equation was derived from the following:

\[EI\phi = \int_{0}^{x} Mdx\] (3.2)

Equation 3.1 yields a displacement of 0.008 inches in the positive x direction at Node A for the loading conditions given in Figure 3.24.

The boundary conditions shown in Figure 3.25 were used to constrain the body.
The finite element analysis of the all-hexahedral mesh resulted in a displacement of 0.006383 inches in the positive x direction at Node A. The finite element analysis mesh is stiffer, causing a smaller displacement than the closed-form solution of 0.008 inches. One explanation for this is that the finite element method generally results in a stiffness matrix which is stiffer than the actual object that is being represented. Thus, smaller displacements are predicted. Another explanation for the smaller displacement is that a linear finite element (such as the eight-node hexahedron) does not model bending displacement states correctly. This phenomenon is called interpolation failure. Figure 3.26 illustrates the difference between what physically happens when bending occurs and how a linear element represents bending.
The "keystoning" shape represented by the finite element in Figure 3.26 is due to interpolation failure. Interpolation failure is the cause of locking.\textsuperscript{17} Locking is a condition of excessive stiffness for a particular deformation state. This situation explains the excessive stiffness and in turn smaller displacement for this finite element analysis of the all-hexahedral mesh above. One approach to minimize the effects of locking which will be explained and employed later in the knife element is reduced integration.

After the finite element analysis was run on the all-hexahedral mesh, a similar run was done using the mesh with two embedded knives. The same loading conditions and constraints (Figure 3.24 and Figure 3.25) were used in the analysis. The mesh with the two knives produced a displacement of 0.006365 inches at Node A as compared to the 0.006383 inches of the all-hexahedral mesh. Since both the mesh containing the two
knives and the all-hexahedral mesh are not symmetric throughout the interior of the mesh, bending loads were applied in six different ways to test for knife-directionality effects. Two loading configurations were applied in each of the three coordinate axis directions (x, y, and z.) The first loading condition described above in Figure 3.24 was in the x direction. This loading condition produced a bending moment around the z axis. The next loading condition in the x direction is shown in Figure 3.27 (the mesh with the two knives is shown.) This loading condition produces a bending moment around the y axis.

![Figure 3.27 - Loading in X direction for bending.](image)

Similar loading conditions were applied in the y direction to produce bending moments around the z and x axes respectively. The same was done in the z direction to produce bending moments around the x and y axes respectively. In this way, the bending moments were applied to the two embedded knives in different directions to check for
orientation effects. The displacements at Node A were calculated for each of these loading conditions applied to the embedded knife mesh and are tabulated below in Table 3-9.

<table>
<thead>
<tr>
<th>Loading Condition</th>
<th>Displacements (inches)</th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X Direction</td>
<td>Y Direction</td>
<td>Z Direction</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Closed Form Solution</td>
<td>0.008</td>
<td>0.008</td>
<td>0.008</td>
<td>0.008</td>
<td>0.008</td>
</tr>
<tr>
<td>All-Hexahedral Mesh</td>
<td>0.006383</td>
<td>0.006922</td>
<td>0.006434</td>
<td>0.007447</td>
<td>0.007839</td>
</tr>
<tr>
<td>Two-Knife Mesh</td>
<td>0.006365</td>
<td>0.006646</td>
<td>0.006697</td>
<td>0.006687</td>
<td>0.006741</td>
</tr>
</tbody>
</table>

The results in Table 3-9 indicate that there are changes in the displacements at Node A due to the orientation of the knives that are embedded in the mesh. This is clear because the displacements are different for different loading orientations. It is also clear from the results that under bending conditions, the knife element does not perform without knife-hex deviation. The average knife-hex deviation between the all-hexahedral mesh and the mesh containing knife elements was about 6.6%.

Earlier, the effects of interpolation failure were discussed. It is likely that the poor performance of the knife is due to this phenomenon. Reduced integration is a method used to counteract the effects of interpolation failure. Reduced integration means using a reduced number of gauss points when doing Gaussian integration. Note that since the components of the stiffness matrix of the knife elements are not integrated exactly with a 6x6x6 Gaussian quadrature, even a 2x2x2 quadrature is reduced. Often times when interpolation failure occurs, spurious shear stresses are introduced into the finite element solution causing excessive stiffness. In many cases, by using a reduced number of gauss points, these spurious effects are missed in the numerical integration. This can produce
more accurate results in some instances. Reduced integration was employed in the knife element to see if it would improve the performance of the knife in bending loading conditions. The hexahedral elements surrounding the knife elements were integrated using the usual 4x4x4 point integration. However, the two knife elements were integrated using 1x1x1 point integration. Table 3-12 shows the results of using reduced integration on the knife element.

<table>
<thead>
<tr>
<th>Loading Condition</th>
<th>X Direction</th>
<th>Y Direction</th>
<th>Z Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>4x4 Knife Integration:</td>
<td>Z Moment</td>
<td>Y Moment</td>
<td>Z Moment</td>
</tr>
<tr>
<td>1x1 Knife Integration:</td>
<td>0.006365</td>
<td>0.006464</td>
<td>0.006697</td>
</tr>
<tr>
<td>All-Hexahedral Mesh:</td>
<td>0.006383</td>
<td>0.006922</td>
<td>0.006434</td>
</tr>
</tbody>
</table>

In each case, using reduced integration improved the performance of the knife element in bending. In some cases, the reduced integration caused the knife mesh to produce results closer to the closed form solution of 0.008 inches than the all-hexahedral mesh.

Global Evaluation (Twenty Knife Elements in Eighty Hex Elements)

Next, a more global evaluation was done. A much larger mesh in the shape of a cantilever beam was constructed and is shown below in Figure 3.28.
For comparison purposes, an all-hexahedral mesh with the same shape was also constructed and is shown below in Figure 3.29.
A bending situation was simulated by applying loads to the mesh as shown in Figure 3.30.
All of the nodes on the left end of the beam were fixed in the three coordinate directions. Displacements for all of the nodes were calculated. However, only node "A" (circled) will be used to compare displacement results for the knife mesh and all-hexahedral mesh. This node was chosen because it is the area in the mesh where the largest displacements occur.

A close-form solution for the displacement in the negative z direction of the end of a cantilever beam is: \(^{16}\)

\[
\delta = \frac{ML^2}{2EI}.
\]  (3.1)

Equation 3.1 yields a displacement of 0.8 inches in the negative z direction at Node A for the loading conditions given in Figure 3.30. The finite element analysis of the all-
hexahedral mesh resulted in a displacement of 0.699369 inches in the negative z direction at Node A.

After the finite element analysis was run on the all-hexahedral mesh, a similar run was done using the mesh with the twenty embedded knives. The same loading conditions and constraints were used in the analysis. The mesh with the twenty knives produced a displacement of 0.658657 inches at Node A as compared to the 0.699369 inches of the all-hexahedral mesh. Thus, the percent knife-hex deviation in using the mesh with twenty knives instead of the mesh with no knives is 5.8%.

Next, two other meshes were constructed. One mesh had only eight knives and the other had only four knives as shown in Figure 3.31.

![Figure 3.31 - Four and eight knife meshes.](image-url)
The same loading conditions and constraints were used for the eight and four knife meshes. The respective displacements are shown in Table 3-11.

<table>
<thead>
<tr>
<th>Mesh Type</th>
<th>Displ. (inches)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>20 Knife Mesh</td>
<td>-0.65866</td>
<td>5.8</td>
</tr>
<tr>
<td>8 Knife Mesh</td>
<td>-0.68029</td>
<td>2.7</td>
</tr>
<tr>
<td>4 Knife Mesh</td>
<td>-0.68983</td>
<td>1.4</td>
</tr>
<tr>
<td>All-Hex Mesh</td>
<td>-0.69937</td>
<td>0.0</td>
</tr>
</tbody>
</table>

It is evident from the results in Table 3-11 that the knife-hex deviation in bending conditions decreases as the volume of the knife elements compared to the total volume of the mesh decreases. Thus, the fewer the knife elements, the smaller the knife-hex deviation in the analysis.

Next, reduced integration (1x1x1) was used on the knife elements in the above meshes and the displacements were recalculated. The results are shown in Table 3-12.

<table>
<thead>
<tr>
<th>Mesh Type</th>
<th>4x4x4 Knife Integration</th>
<th>1x1x1 Knife Integration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Disp. (inches)</td>
<td>Disp. (inches)</td>
<td></td>
</tr>
<tr>
<td>20 Knife Mesh</td>
<td>-0.65866</td>
<td>-0.77668</td>
</tr>
<tr>
<td>8 Knife Mesh</td>
<td>-0.68029</td>
<td>-0.73222</td>
</tr>
<tr>
<td>4 Knife Mesh</td>
<td>-0.68983</td>
<td>-0.71579</td>
</tr>
</tbody>
</table>

It is interesting to note that when using reduced integration, the displacements of the meshes with knife elements were even closer to the closed-form solution of 0.8 inches than the all-hexahedral mesh. Also, as fewer knife elements are used in the mesh, the displacements get further and further away from the closed-form solution indicating that the knife elements integrated with reduced integration contribute to the flexibility of the mesh and the more of these elements you use, the more flexible the mesh.
Bending Stress vs. Axial Stress

From the patch test, it was shown that under axial loading, the knife element did not produce any knife-hex deviation. However, in bending, there was knife-hex deviation caused by the knife element being stiffer than hexahedral elements. The goal here is to quantify how the knife-hex deviation increases or decreases depending on how much bending there is compared to axial force.

The strategy was to start off with a bending load applied to a mesh, analogous to the previous examples in the Localized Evaluation section (2 Knife/8 Hex Mesh,) and then add an axial force and gradually increase the magnitude of the axial force to determine if the knife-hex deviation from the bending would decrease as the axial force dominated the loading conditions. The displacements due to both bending and axial forces were calculated each time more axial force was added using finite element methods and closed form solutions. Figure 3.32 shows a plot of the calculated data. The loading conditions in the x direction causing a moment around the z axis (see Figure 3.24) were arbitrarily chosen for this change in knife-hex deviation evaluation.
Figure 3.32 - Plot of stress ratio vs. percent knife-hex deviation.

For each step, a ratio of axial stress to bending stress was plotted against the percent knife-hex deviation between the finite element results and the closed form results. As would be expected, as the ratio of axial stress to bending stress increased, the knife-hex deviation decreased.

Summary of Bending

The knife element was embedded in a mesh of hexahedral elements in order to subject it to bending moments. Displacements were calculated based on the bending loads that were applied. It was suggested that linear elements such as the eight-node hexahedron do not interpolate the deformation state of bending adequately because of interpolation failure. It was also suggested that the knife element suffers from the same
phenomena and thus produces excessive stiffness under bending loads. Reduced integration was employed in integrating the knife element to counteract the effects of interpolation failure. It was shown that reduced integration improved the performance of the knife element under bending conditions. Using $4\times4\times4$ Gaussian integration, the knife meshes produced displacements with knife-hex deviation on the average of about 6% of the displacements from the all-hexahedral mesh. Using $1\times1\times1$ reduced integration, the knife mesh produced larger displacements that were sometimes closer to the closed-form solution than the all-hexahedral mesh. As loading conditions changed from pure bending to pure axial, the knife-hex deviation decreased. This phenomenon was plotted.
Chapter 4

4. Conclusions and Usage Requirements

The knife shape and connectivity are artifacts of current automatic all-hexahedral meshing algorithms. Because the invalid connectivity of the knife is not compatible with current analysis packages, various solutions have been considered for dealing with the problem. The pros and cons of each are discussed below. Then knife element usage requirements are prescribed.

4.1 Pros and Cons of Solutions to Knife Connectivity

4.1.1 Knife Propagation

One alternative is to propagate the knife out to the surface of the mesh or to another knife where the knife connectivity can be resolved. Even though resolving the knife connectivity would be ideal, there are some drawbacks to this alternative. In order to resolve the knife connectivity with another knife, another knife must always exist and must be able to be propagated to the location of the first knife in order to join. Also, if the knife is to be driven to the surface to resolve the knife connectivity, the surface mesh will be altered. Often times the surface mesh is specified such that it cannot be altered because of boundary conditions. Another problem occurs when two volumes join at a surface and are meshed independently of each other. The surface mesh of the one volume must be forced to match that of the other volume and therefore one of the volumes cannot
alter the joining surface mesh because of knife propagation. Because of these drawbacks
to knife propagation, it is not a totally viable solution to the knife connectivity.

4.1.2 Manually Changing the Knife Connectivity

Another solution to the knife connectivity is to manually adjust the mesh after an
automatic mesh generator has finished. This is a tedious process and can be quite time
consuming. Also, the reason automatic mesh generators are being developed is to
eliminate the need to create meshes by hand.

4.1.3 Filling the Knife Region with Tetrahedral Elements

Filling the knife region with tetrahedral elements isn't an option unless a
transition element is used to join the quadrilateral faces of the neighboring hexahedral
elements to the triangular faces of the tetrahedral elements. In the case that transition
elements are used, the tetrahedral elements are stiffer than the knife element and therefore
result in poorer solutions as was shown earlier in this thesis.

4.1.4 Leave the Knife Connectivity in the Mesh

The final solution is to leave the knife connectivity in the mesh and use the knife
finite element that was developed in this thesis. This solution requires no extra work
once the knife element is implemented into the analysis code. Leaving the knife in the
mesh will not disrupt the mesh anywhere else and will not require transition elements.
Since the knife element is less stiff than tetrahedral elements, it will produce more
accurate results than meshes containing both hexahedra and tetrahedra.
4.1.5 Conclusion

It is concluded from the evaluation done on the knife finite element, that leaving the knife connectivity in the mesh and using the knife finite element is an acceptable solution to the knife connectivity problem.

4.2 Usage Requirements

In pure axial loading conditions, the knife finite element can be used without any knife-hex deviation.

In loading conditions where bending is present, care should be taken in using the knife finite element. In the immediate location of the knife element, knife-hex deviations on the average of 6% were observed under pure bending loads. As the ratio of axial stress to bending stress increases, the knife-hex deviation drops considerably as seen in Figure 3.32. Therefore, if knife connectivity arises in a mesh and the analysis reveals high stress gradients in the area of the knife element, efforts can be made to propagate the knife to a low stress gradient area. However, the knife-hex deviation caused by the knife element decreases quickly as you move away from the knife in the mesh. Therefore, if the knife is not near an area where high accuracy (such as on a surface) is required, the knife can be left embedded.
References

The Development and Evaluation of the

Knife Finite Element

Brett W. Clark

Department of Mechanical Engineering

M.S. Degree, August 1996

ABSTRACT

This thesis presents the development and evaluation of the knife finite element which is a degenerate case of a hexahedral element. The knife connectivity is an artifact of automatic all-hexahedral mesh generators. Currently, knives are propagated to the surface of the mesh for removal. However, since this disturbs the surface mesh, other alternatives are needed. This thesis investigates the option of leaving the knife connectivity in the mesh and treating it as a valid finite element. The shape functions and stiffness matrix for the knife element are derived and evaluated using theoretical and practical evaluations.

It is concluded that the knife finite element is a viable element and should be used in finite element analysis when the knife connectivity occurs. Using the knife element reduces the work involved with fixing the knife connectivity by propagation or other means and will produce acceptable results in most cases.

COMMITTEE APPROVAL:

Steven E. Benzley, Committee Chair

Jordan J. Cox, Committee Member

Carl D. Sorensen, Committee Member

Brent W. Webb, Graduate Coordinator