Title: Verification of the W76-1 Hostile Environments Model

Author(s): R. Robert Stevens, ESA-WR

Submitted to: 2nd Annual Nuclear Engineering Conference
Monterey, CA
September 4-6, 2002
ABSTRACT:

Demonstrating mesh convergence for a finite element analysis requires multiple meshes, but creating high quality meshes is a time-consuming task. Furthermore, estimates of the amount of error caused by mesh refinement are difficult to make for a sequence of unrelated, unstructured finite element meshes. A solution for both of these problems is to automatically generate a refined mesh by subdividing every element in the original mesh. The resulting refined mesh has a uniform "mesh refinement ratio" (relative to the original mesh), so established mesh convergence error estimators, such as Roache's Grid Convergence Indicator (GCI), can be applied.

This presentation will cover the process of automatically generating a refined mesh, and discuss the Grid Convergence Indicator (GCI) error metric. The GCI will be applied to two models subjected to transient loadings: a simple test problem and a high-fidelity model of an unclassified W76 component. The mesh convergence exhibited by the analysis code DYNA3D will be discussed.
Verification of the W76-1 Hostile Environments Model

R. Robert Stevens
ESA-WR, LANL

Presented at the Second Annual Nuclear Weapons Engineering Analysis Conference
Monterey, California
September 4-6, 2002
Verification of the W76-1 Hostile Environments Model

Statement of the Problem:
Demonstrating mesh convergence requires multiple meshes. This meshing can be a time-consuming task.

Estimating the error from mesh convergence for UNRELATED, unstructured meshes is not well-established.

A solution for both of these problems is to automatically generate a refined mesh by subdividing an original mesh.

Established methods of estimating the error caused by lack of mesh convergence can then be used.

R. Robert Stevens
ESA-WR, LANL
Nuclear Weapons Engineering Conference
Monterey, CA
4-Sept-2002
Related vs. Unrelated Meshes

Original Mesh

A refined but unrelated mesh of the same part.

Every element of the original mesh subdivided into 4 new elements.

\[ r = \left( \frac{N_{\text{fine}}}{N_{\text{coarse}}} \right)^{1/D} \]

\[ r = 2 \]
Automatic Mesh Subdivision

Developed a computational tool to automatically subdivide every element in a mesh:
- brick, wedge: 1->8
- quad, tri: 1->4

Allows relatively easy method to verify mesh convergence.

Can be applied to entire mesh or to selected parts only.

Reads and writes I-DEAS universal files. Handles groups (node sets and element sets).
Element Subdivision
Kernel = 1x1x1
Element Subdivision

Kernel = 3x3x3

Original Mesh

1 Subdivision

2 Subdivisions

3 Subdivisions
Pressure Applied on Back Face of Box (transient)

The two parts are initially touching.

The contact conditions allow sliding and separation. (Note: no separation actually occurs.)

The strains and relative displacement are small.

Analysis code: Dyna3d (Explicit FEM)

Response metrics:
- Y-displacement at a point on the hanger
- Strain at a point on the hanger
- Box Y-momentum
- Box Z-momentum
- Whole-model kinetic energy
The assumed form of convergence is:

\[ f = A h^p + C \]

where:
- \( f \) = Solution variable
- \( h \) = element edge length (measure of mesh refinement)
- \( A, C, p \): fitting constants

The value \( p \) is the polynomial order of convergence of the mesh refinement.

\( f \) may be any solution variable or "functional" (combination of many "direct" solution variables):
- peak nodal displacement
- momentum
- energy
- stress or strain

The solution variable "\( f \)" for each mesh must be compared at the same point on the part. This can be difficult for integration-point variables.
Convergence of Several Response Metrics

**Test Problem**

1. **Peak Y-Displacement at point on Hanger:** $p = 1.7$

2. **Whole-Model Peak Kinetic Energy:** $p = 1.5$

3. **Box Peak Y-momentum:** $p = 1.2$

4. **Box Peak Z-momentum**
Convergence of Strain Metric

Due to a nonlinear strain gradient at this location, the method of averaging the strains at the neighboring strain gage elements introduces a new source of error.

If the 2x2x2 and 4x4x4 kernel meshes are ignored (because their strain gage integration points are too far away from the response point), the strain metric converges (approximately) linearly.

This mesh had an integration point at the response point; all other meshes had a node at the response point, and used an average of the 4 “connected” strain gage elements.

Extrapolated fine mesh value = -0.0081383

Peak value of stress (X-direction) near base of hanger
Grid Convergence Index (GCI)

The Grid Convergence Index (GCI) is a measure of convergence-related error that takes into account the polynomial order of convergence of the numerical method.

Let $\varepsilon$ be the fractional change in solution between two meshes:

$$\varepsilon = \frac{f_1 - f_2}{f_2}$$

$f_1$ = coarse-mesh solution
$f_2$ = fine-mesh solution

The GCI uses the correction to $f_2$ from the Richardson extrapolation as an error estimator, and also introduces a “factor of safety:”

$$f_{exact} = f_2 + \frac{f_1 - f_2}{r^p - 1}$$

$r$ = mesh refinement ratio
$p$ = polynomial order of convergence

$$GCI_{fine} = F_s \frac{|\varepsilon|}{r^p - 1}$$

$F_s$ = factor of safety

$$GCI_{coarse} = GCI_{fine} + F_s |\varepsilon|$$

For $F_s = 1.0$

Illustration of Grid Convergence Indicator

Example Problem:

1) Two analysis codes:
   code #1: \( p = 3.0 \)
   code #2: \( p = 1.4 \)

2) Two meshes were run with both codes.

3) Epsilon (the fractional change from coarse to fine mesh solutions) is equal for both calculations.

The GCI is much smaller for the higher-order code, properly reflecting the smaller error in the higher-order code’s fine-grid solution.

Note that epsilon is NOT an error estimator!

Equal mesh refinement for each code:
\[ r = \frac{10}{6} = 1.67 \]
The forward mount is the primary load path connecting the major components in the W76/Mk4.

Creating the mesh for this part was very time consuming. Nobody wants to do it over again “just for a mesh convergence study.”
Forward Mount Model Verification: Mesh Details

ORIGINAL MESH
360,000 elements

SUBDIVIDED MESH
2.5 million elements
Forward Mount Model Verification: Displacement at Two Points

Y-direction displacement at the tip of the AF&F. Metric = Peak value.

$$\varepsilon = -1.04\%$$
$$F_s = 1.0$$
$$r = 2$$
$$p = 1.7$$
$$GCI_{coarse} = 1.50\%$$

Y-direction displacement at the 270-degree Forward Mount Tripod Leg.

$$\varepsilon = 3.39\%$$
$$F_s = 1.0$$
$$r = 2$$
$$p = 1.7$$
$$GCI_{coarse} = 4.90\%$$
Forward Mount Model Verification: Stress at Tripod Leg Blend

Location of high stress near blend at forward mount tripod leg.

Effective stress at this location. Used average of 8 "child" bricks for fine mesh.

\[\varepsilon = -4.34\%\]
\[F_s = 1.0\]
\[r = 2\]
\[p = 1.0\]

\[GCI_{coarse} = 8.68\%\]
Table of GCI's for all comparison metrics:

W76 Forward Mount Verification

For all GCI calculations, $F_s = 1.0$, $r = 2.0$.

<table>
<thead>
<tr>
<th>Response Metric</th>
<th>$p$</th>
<th>$\epsilon$</th>
<th>GCI (coarse)</th>
</tr>
</thead>
<tbody>
<tr>
<td>AF&amp;F, Momentum in Y-direction (peak value)</td>
<td>$p = 1.2$</td>
<td>$\epsilon = -1.91%$</td>
<td>92.3 / 94.1</td>
</tr>
<tr>
<td>Potential Energy (AF&amp;F and Forward Mount) (peak value)</td>
<td>$p = 1.5$</td>
<td>$\epsilon = -1.69%$</td>
<td>35.0 / 35.6</td>
</tr>
<tr>
<td>Displacement of AF&amp;F tip, Y-direction (at rebound)</td>
<td>$p = 1.7$</td>
<td>$\epsilon = -2.79%$</td>
<td>-0.396 / -0.383</td>
</tr>
<tr>
<td>Displacement of AF&amp;F tip, X-direction (at rebound)</td>
<td>$p = 1.7$</td>
<td>$\epsilon = -3.79%$</td>
<td>-0.0316 / -0.0306</td>
</tr>
<tr>
<td>Displacement at 270-deg. tripod, Y-direction (peak value)</td>
<td>$p = 1.7$</td>
<td>$\epsilon = -1.05%$</td>
<td>-0.0663 / -0.0670</td>
</tr>
<tr>
<td>Displacement at 270-deg. tripod, X-direction (peak value)</td>
<td>$p = 1.7$</td>
<td>$\epsilon = -3.01%$</td>
<td>-0.0354 / -0.0365</td>
</tr>
<tr>
<td>Effective Stress, strain gage near 270-deg. tripod (peak value). Used average of 4 strain gages on refined model.</td>
<td>$p = 1.0$</td>
<td>$\epsilon = -4.64%$</td>
<td>0.90 / 0.042 MPa</td>
</tr>
<tr>
<td>Effective Stress, near blend at 270-deg. tripod (peak value). Used average of 8 bricks on refined model.</td>
<td>$p = 1.0$</td>
<td>$\epsilon = -4.34%$</td>
<td>75.0 / 78.4 MPa</td>
</tr>
</tbody>
</table>
Model Verification via Sub-Assemblies

A) The mesh for the entire model is not refined: a subdivision of the whole model would be far too big to use. Only single parts or small sub-assemblies, taken in isolation, are “tested.”
  - \( n^4 \) scaling: subdividing bricks yields 8 times as many elements and one-half the stable time step.

B) The actual loading is not used: a substitute load applied to the part or parts in the subassembly is created. The substitute load should “exercise” the subassembly as closely as possible to the way the real load does.

C) All other analysis features remain the same: same analysis code, material models, contact types, etc.

Question: if the structural response at one point is strongly affected by many other parts, then how do mesh convergence error “add up?”

If you refine the mesh of the entire model, you won’t know which parts are the largest contributors to mesh convergence error (i.e., culprit parts).