

LA-UR-02-510

Approved for public release;  
distribution is unlimited.

*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



Los Alamos National Laboratory, an affirmative action/equal opportunity employer, is operated by the University of California for the U.S. Department of Energy under contract W-7405-ENG-36. By acceptance of this article, the publisher recognizes that the U.S. Government retains a nonexclusive, royalty-free license to publish or reproduce the published form of this article, or to allow others to do so, for U.S. Government purposes. Los Alamos National Laboratory requests that the publisher identify this article as work performed under the auspices of the U.S. Department of Energy. Los Alamos National Laboratory strongly supports academic freedom and a researcher's right to publish; as an institution, however, the Laboratory does not endorse the viewpoint of a publication or guarantee its technical correctness.

TITLE: Verification of the W76-1 Hostile Environments Model

AUTHOR: Ralph Robert Stevens

CLASSIFICATION: Unclassified

ORGANIZATION: Engineering Science and Applications (ESA) Division  
Weapon Response (WR) Group  
Los Alamos National Laboratory (LANL)

KEYWORDS: Verification, Mesh Convergence, Element Subdivision, GCI

LAUR NUMBER:

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.

Approved for Unlimited Release  
LA-UR-02-XXXX  
August 8, 2002

UNCLASSIFIED



## Verification of the W76-1 Hostile Environments Model

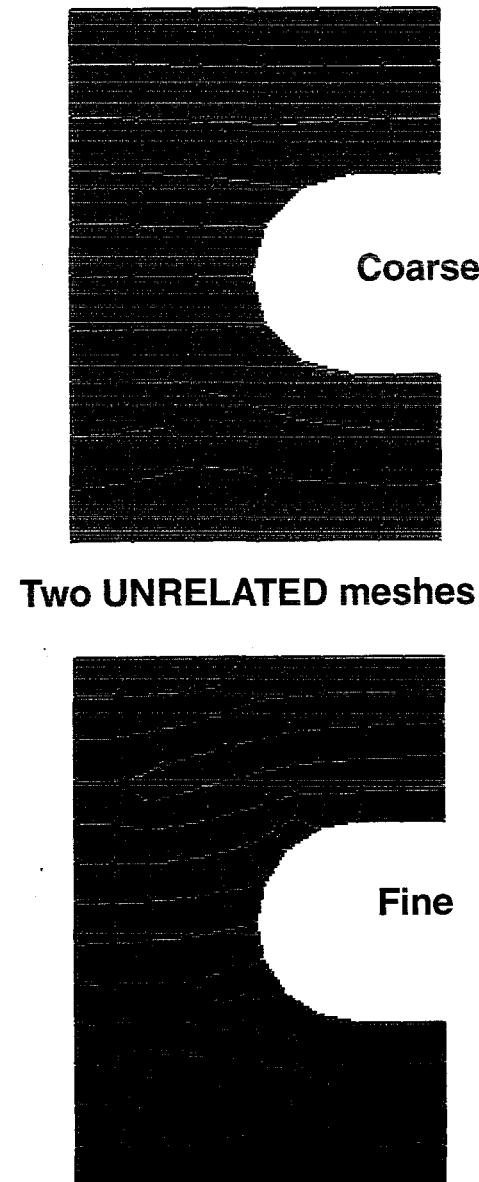
R. Robert Stevens  
ESA-WR, LANL

Presented at the Second Annual Nuclear Weapons Engineering Analysis Conference  
Monterey, California  
September 4-6, 2002

Engineering Science & Applications

UNCLASSIFIED

# Verification of the W76-1 Hostile Environments Model



Two UNRELATED meshes

## 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.

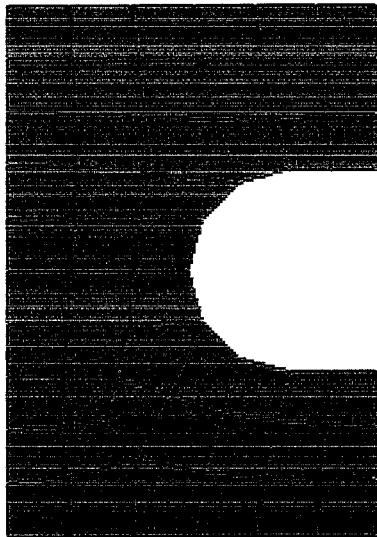
Established methods of estimating the error caused by lack of mesh convergence can then be used.

R. Robert Stevens  
ESA-WR, LANL

*ANALYSIS*  
Nuclear Weapons Engineering Conference  
Monterey, CA  
4-Sept-2002

UNCLASSIFIED

# Related vs. Unrelated Meshes



Original Mesh

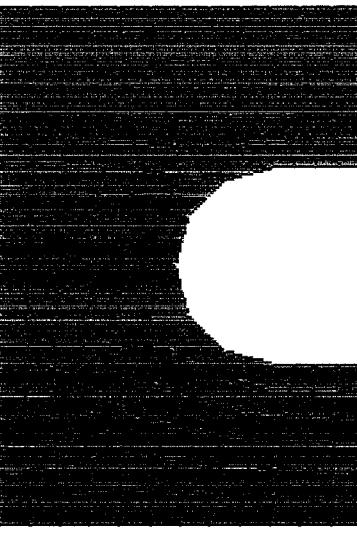
**r = Refinement Ratio:**



A refined but  
unrelated mesh  
of the same part.

$$r \approx \left( \frac{N^{fine}}{N^{coarse}} \right)^{1/D}$$

$r = 2$



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

NOT RECOMMENDED WHEN  
REFINEMENT IS NOT  
STRUCTURED AND FAIRLY

# Automatic Mesh Subdivision

Developed a computational tool to automatically subdivide every element in a mesh:

brick, wedge: 1->8

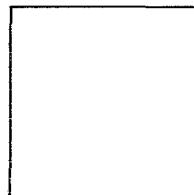
quad, tri: 1->4

Allows relatively easy method to verify mesh convergence.

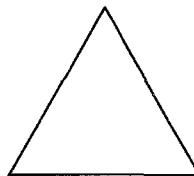
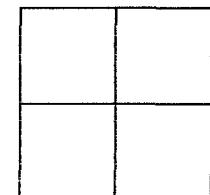
Can be applied to entire mesh or to selected parts only.

Reads and writes I-DEAS universal files. Handles groups (node sets and element sets).

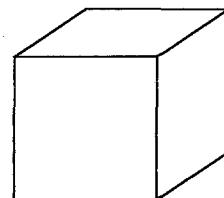
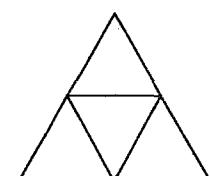
*element  
preserves' aspect ratio and interior  
angles*



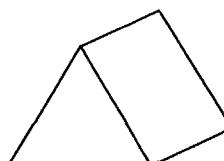
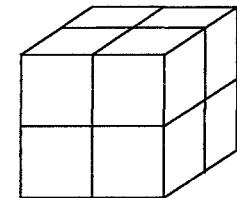
Quad



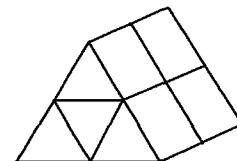
Tri



Brick

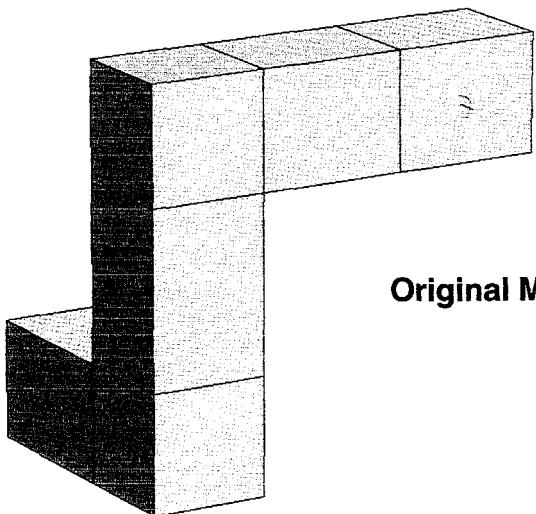


Wedge

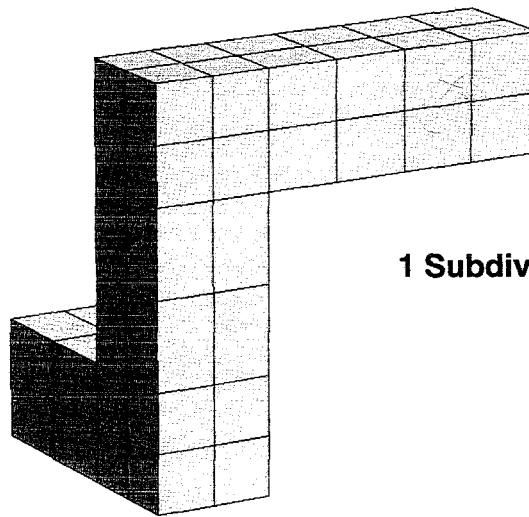


# Element Subdivision

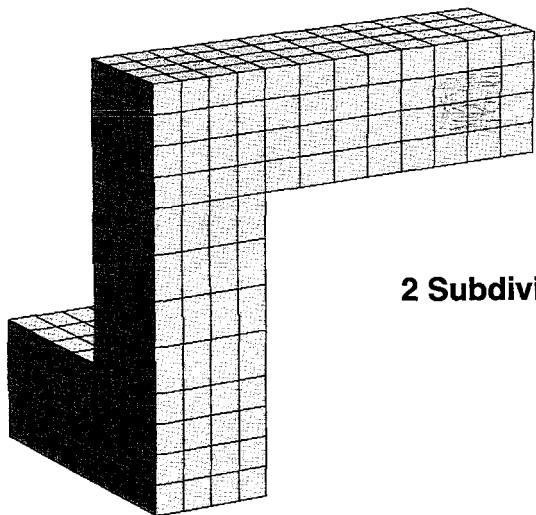
**Kernel = 1x1x1**



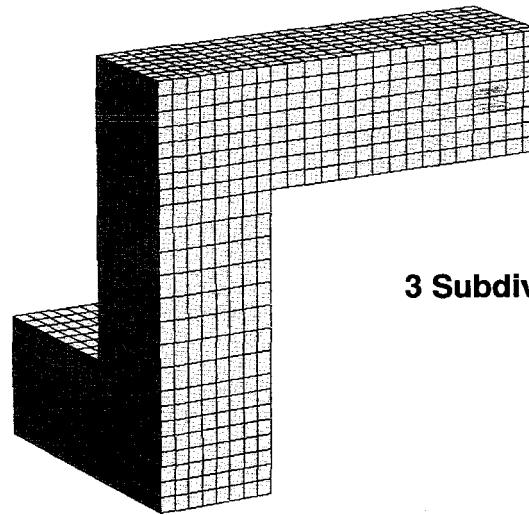
**Original Mesh**



**1 Subdivision**



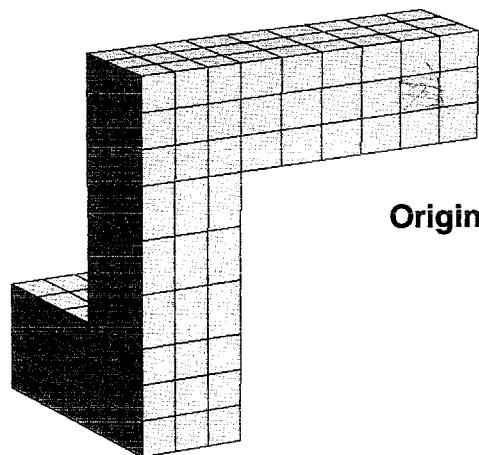
**2 Subdivisions**



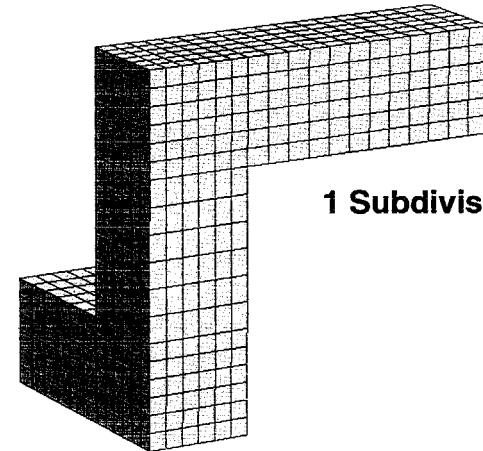
**3 Subdivisions**

# Element Subdivision

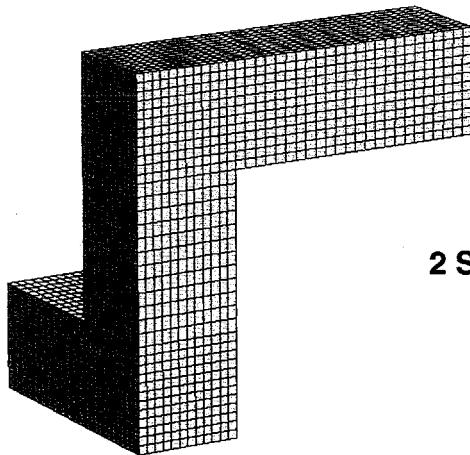
**Kernel = 3x3x3**



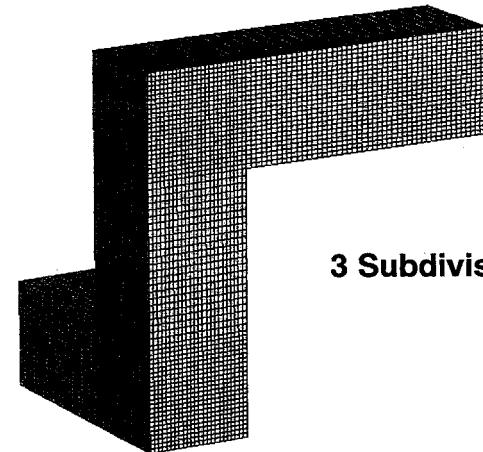
**Original Mesh**



**1 Subdivision**

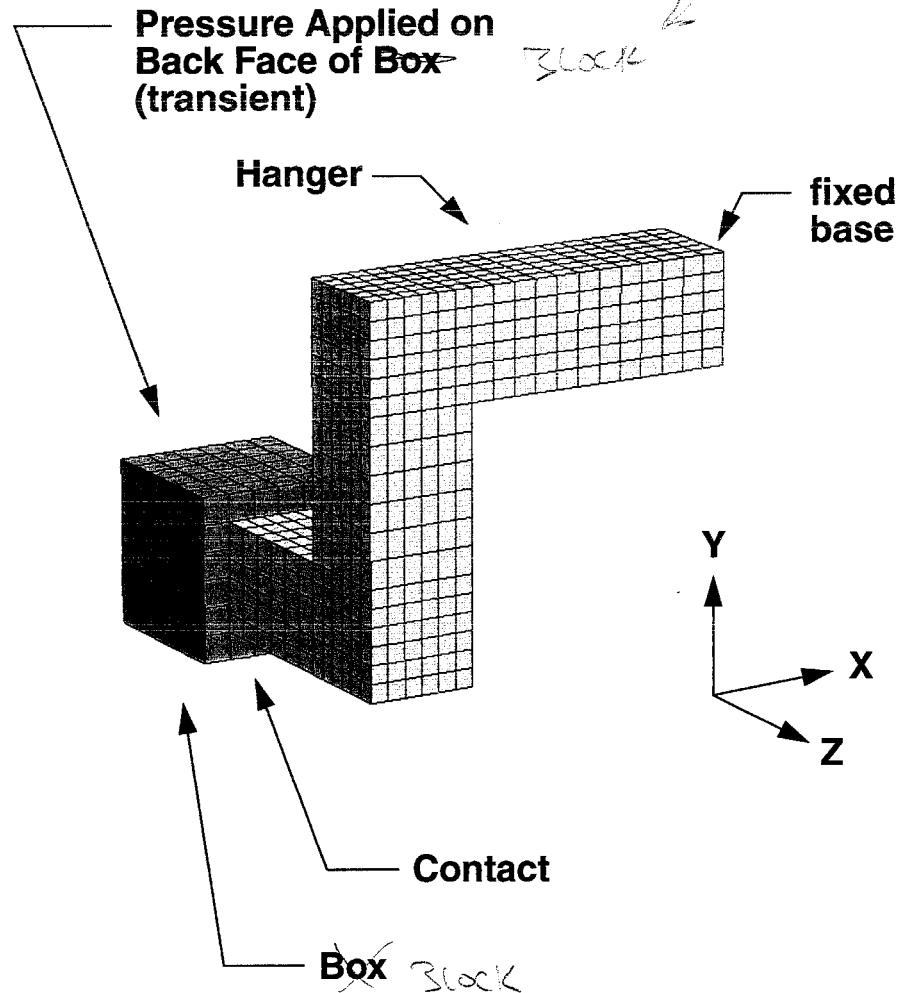


**2 Subdivisions**



**3 Subdivisions**

# Test Problem



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

# Convergence with Mesh Refinement

The assumed form of convergence is:

$$f = A h^p + C$$

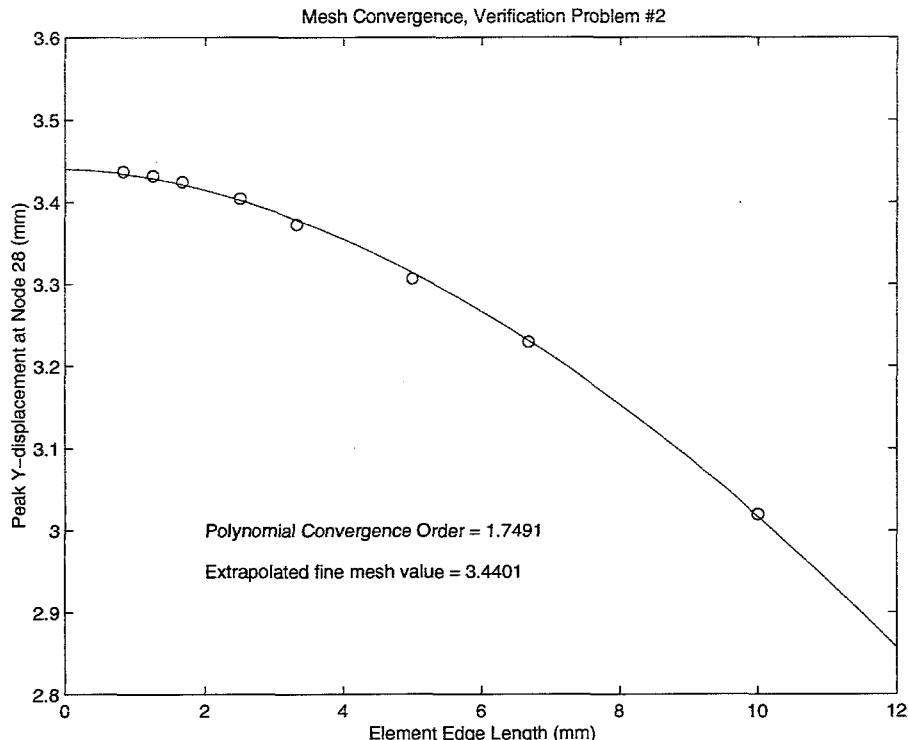
where:

$f$  = 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):

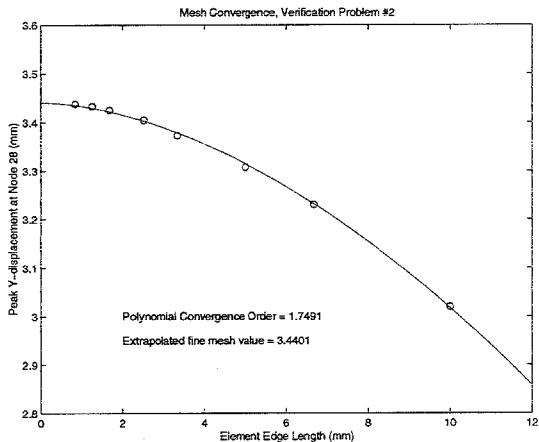
- peak nodal displacement
- momentum
- energy
- stress or strain



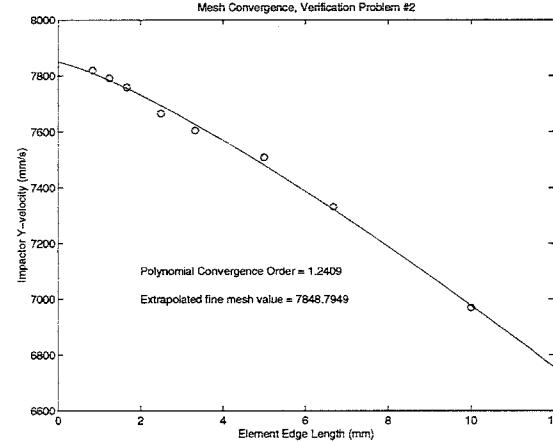
The solution variable “ $f$ ” for each mesh must be compared at the same point on the part. This can be difficult for integration-point variables.

# Convergence of Several Response Metrics

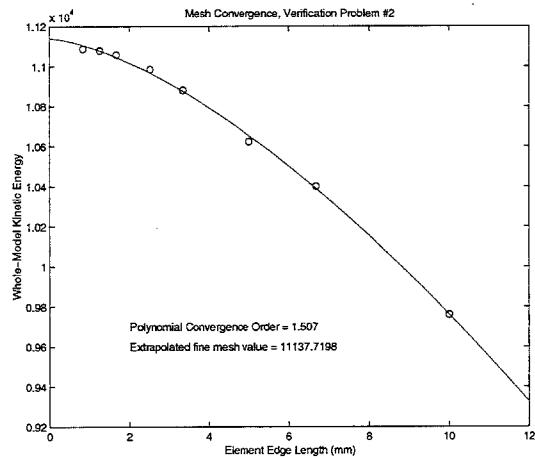
## Test Problem



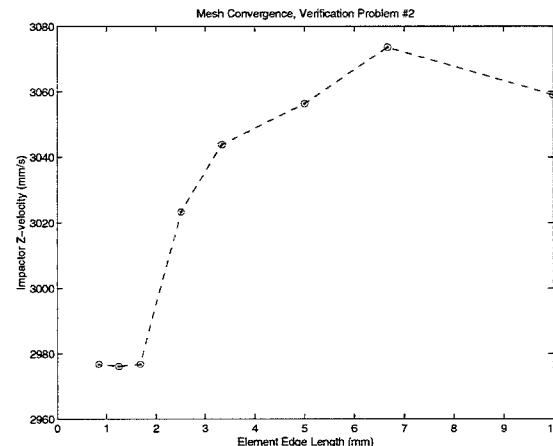
Peak Y-Displacement at point on Hanger:  $p = 1.7$



Box Peak Y-momentum:  $p = 1.2$   
Block



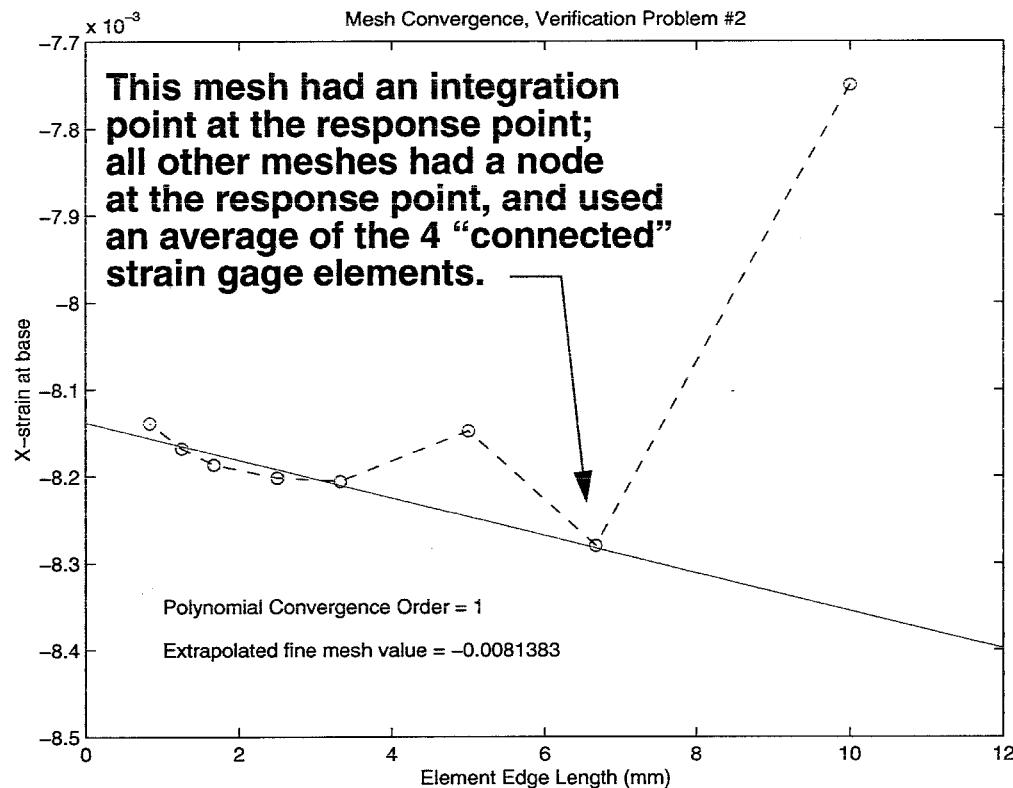
Whole-Model Peak Kinetic Energy:  $p = 1.5$



Box Peak Z-momentum

Block

# Convergence of Strain Metric



*Varying*  
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.

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

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

$f_1$  = 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:”

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

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

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

$F_s$  = factor of safety

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

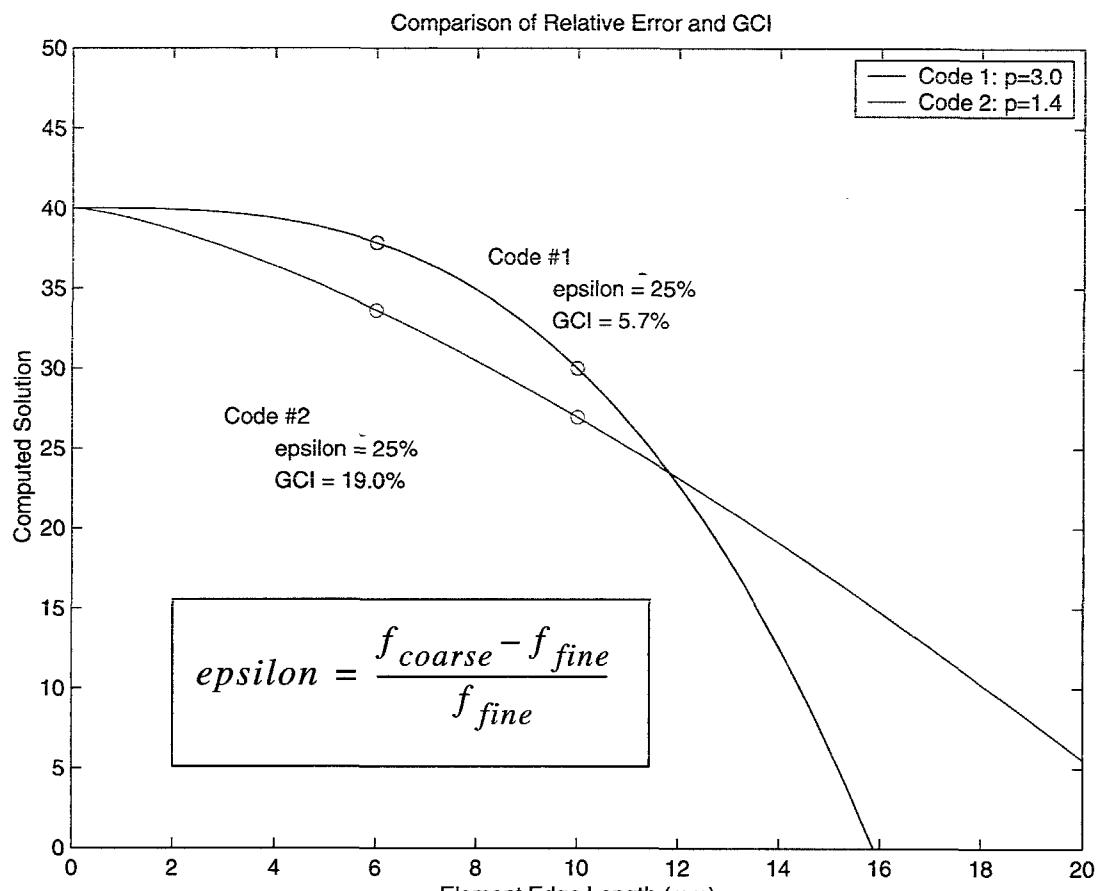
$$GCI_{fine} \equiv \left| \frac{f_{exact} - f_2}{f_2} \right|$$

$\approx \frac{f_{exact} - f_2}{f_2 - 1}$

For  $F_s = 1.0$

Ref.: Patrick Roache, “Verification and Validation in Computational Science,” 1998

# Illustration of Grid Convergence Indicator



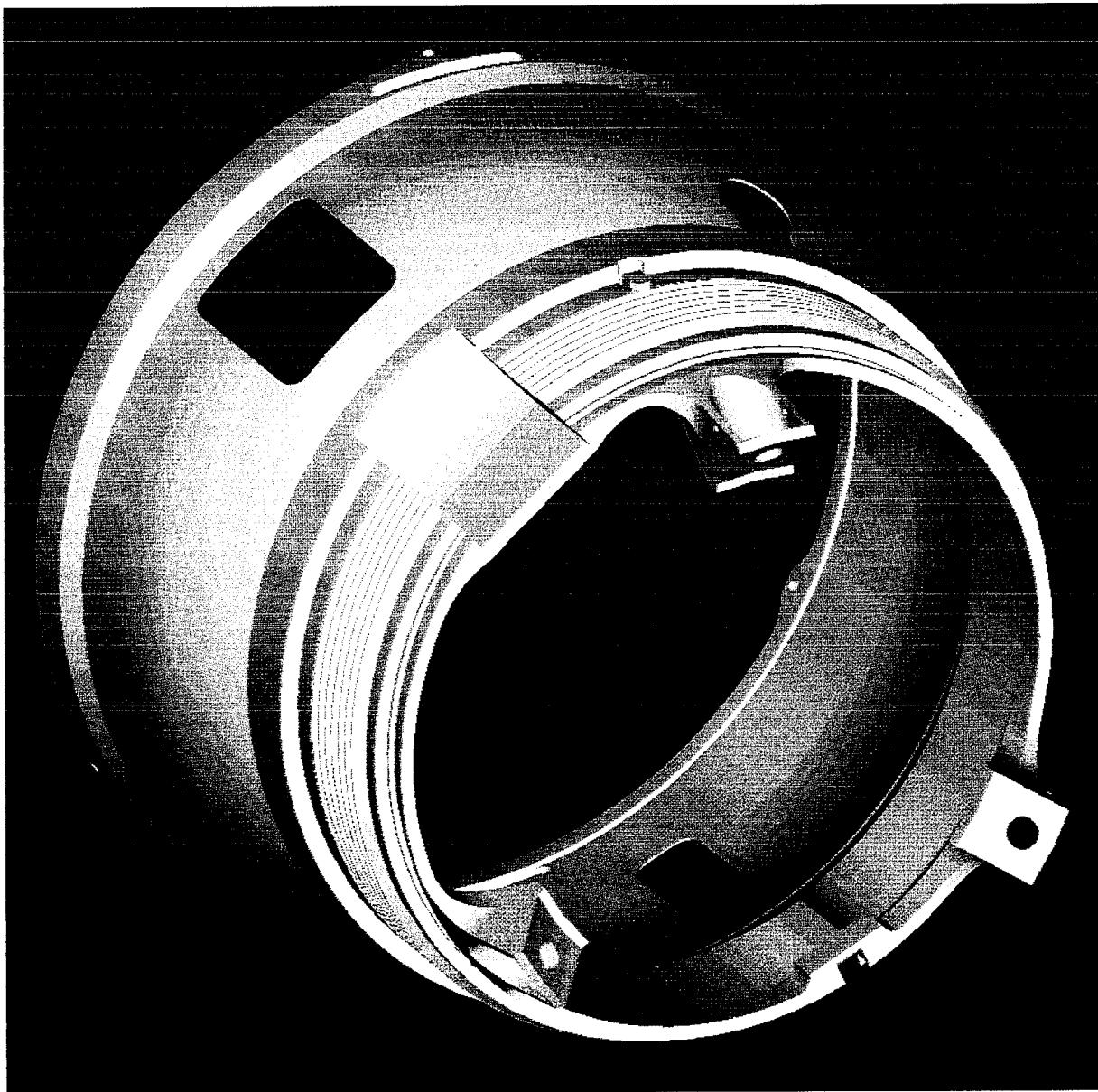
Equal mesh refinement for each code:  
 $r = 10/6 = 1.67$

## 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!

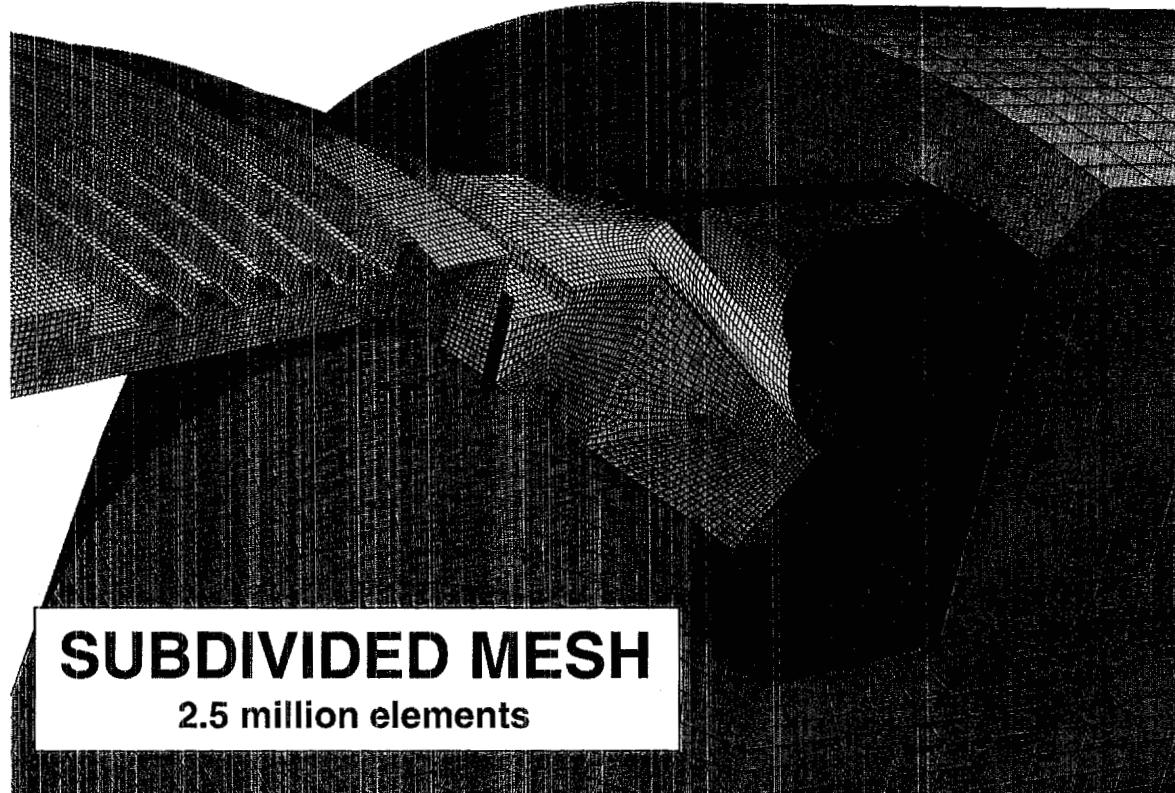
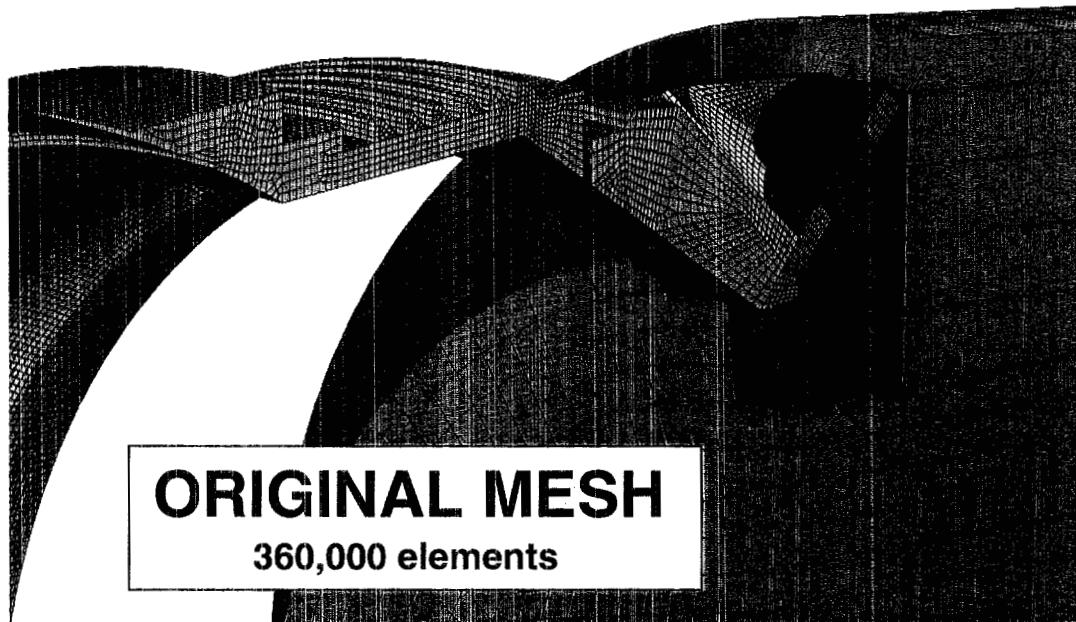


## **Forward Mount Model Verification**

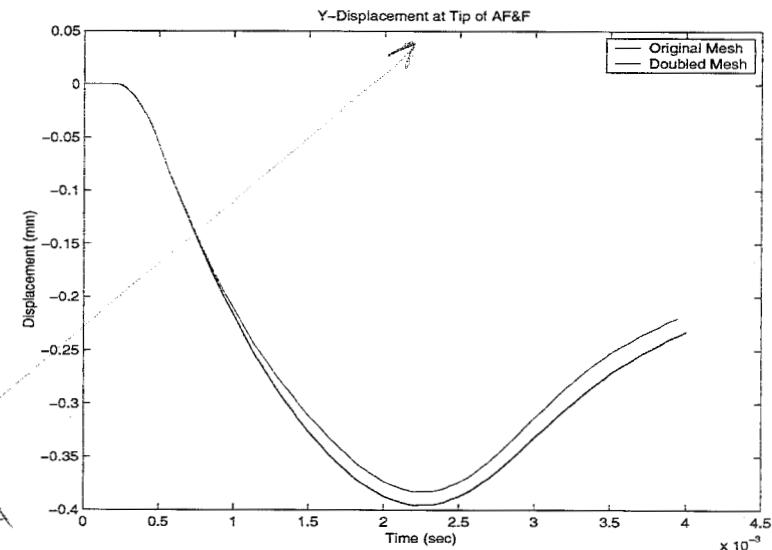
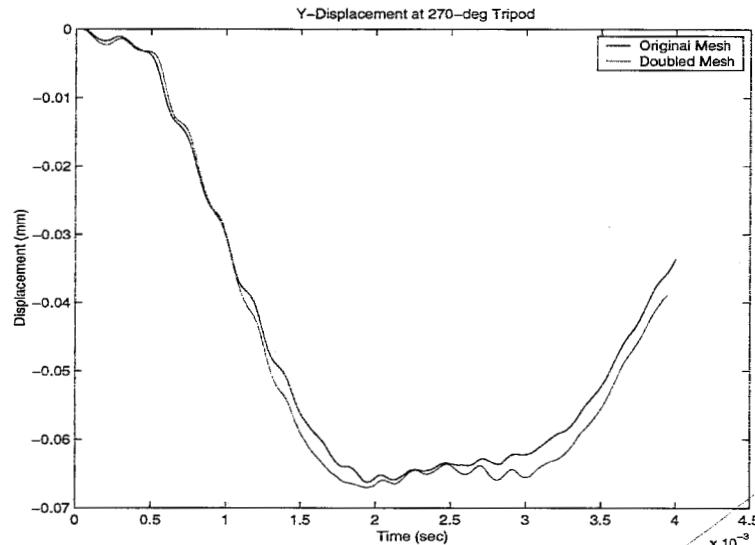
**The forward mount is the primary load path connecting the major components in the W76 / Mk4.**

**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



# Forward Mount Model Verification: Displacement at Two Points



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

$$\varepsilon = -1.04\%$$

$$F_s = 1.0$$

$$r = 2$$

$$p = 1.7$$

$$GCI_{coarse} = 1.50\% \quad (1)$$

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

$$\varepsilon = 3.39\%$$

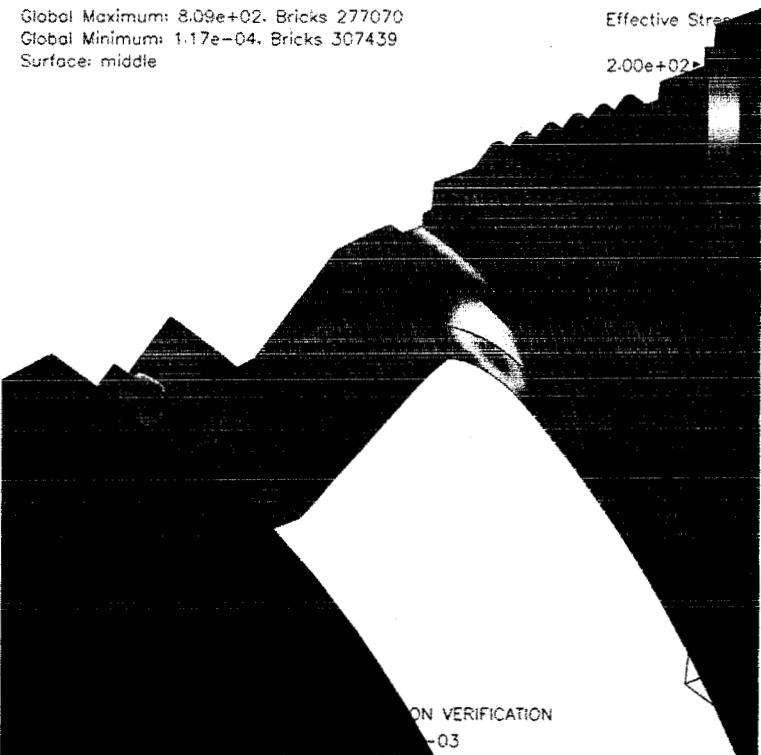
$$F_s = 1.0$$

$$r = 2$$

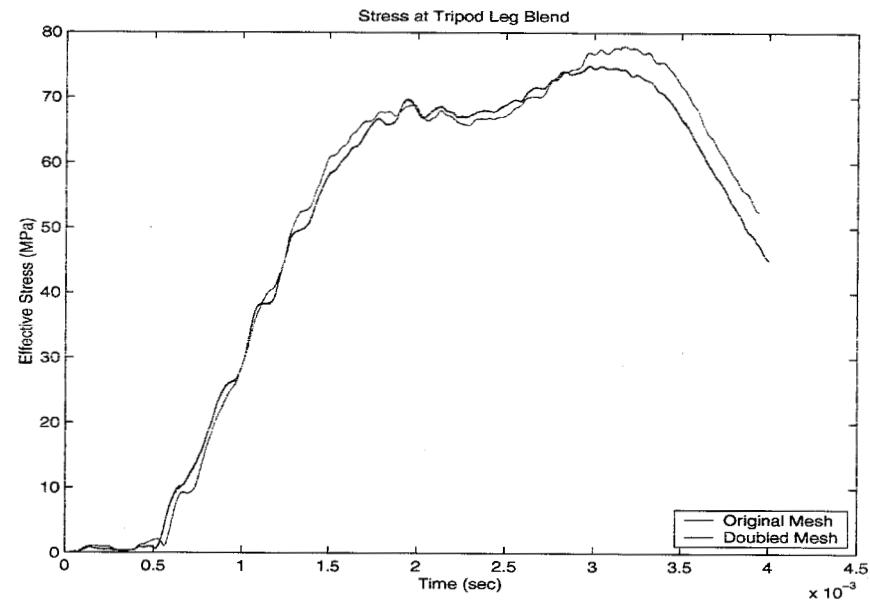
$$p = 1.7$$

$$GCI_{coarse} = 4.90\%$$

# 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,  $F_s = 1.0$ ,  $r = 2.0$ .

Response Metric	$p$	GCI (coarse)
AF&F, Momentum in Y-direction (peak value)	$P = 1.2$ $\epsilon = -1.91\%$	92.3 / 94.1
Potential Energy (AF&F and Forward Mount) (peak value)	$P = 1.5$ $\epsilon = -1.69\%$	35.0 / 35.6
Displacement of AF&F tip, Y-direction (at rebound)	$P = 1.7$ $\epsilon = +3.39\%$	-0.396 / -0.383
Displacement of AF&F tip, X-direction (at rebound)	$P = 1.7$ $\epsilon = +3.28\%$	-0.0316 / -0.0306
Displacement at 270-deg. tripod, Y-direction (peak value)	$P = 1.7$ $\epsilon = -1.05\%$	-0.0663 / -0.0670
Displacement at 270-deg. tripod, X-direction (peak value)	$P = 1.7$ $\epsilon = -3.01\%$	-0.0354 / -0.0365
Effective Stress, strain gage near 270-deg. tripod (peak value). Used average of 4 strain gages on refined model.	$P = 1.0$ $\epsilon = -4.86\%$	0.90 / 0.042 MPa 0.942
Effective Stress, near blend at 270-deg. tripod (peak value). Used average of 8 bricks on refined model.	$P = 1.0$ $\epsilon = -4.34\%$	75.0 / 78.4 MPa

GCI = 3.28%

2.61%

4.90

4.74

1.51

4.35

?

8.92%

8.67

# 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).**