Structural Integrity Associates, Inc.

CALCULATION PACKAGE

File No.: 0801040.301

Project No.: 0801040

Quality Program: ☒ Nuclear ☐ Commercial

PROJECT NAME:
Monticello Steam Dryer Submodel Analysis

CONTRACT NO.:
08-510

CLIENT:
Continuum Dynamics, Inc.

PLANT:
Monticello Nuclear Generating Plant

CALCULATION TITLE:
Steam Dryer Outer Hood Submodel Analysis

Document Revision	Affected Pages	Revision Description	Project Manager Approval Signature & Date	Preparer(s) & Checker(s) Signatures & Date
0	1 - 30 A-1 - A-2	Initial Issue	Karen K. Fujikawa Karen K. Fujikawa 10/31/08	Minghao Qin Minghao Qin 10/31/08 Kok Soo Bee Soo Bee Kok 10/31/08

~~Contains Vendor Proprietary Information~~

Table of Contents

~~Contains Vendor Proprietary Information~~

List of Tables

List of Figures

1.0 INTRODUCTION

Background

The Monticello steam dryer has been analyzed using a shell finite element model, which does not specifically model the weld. The results show that there are localized high stresses at some of the welded connections. Based on past experience, it has been established that if the connections are modeled using solid elements to represent the weld, the weld can better distribute the stresses in the region of high stress concentration. Coupled with the use of stress linearization approach, the maximum stress intensity computed using the solid model (with the weld modeled) generally gives a better prediction of stresses compared with shell elements in the region of the high stress concentration.

Objective

This calculation documents the comparison of the stress intensity computed using a shell finite element model and a solid finite element model with linearization stress paths in the region of interest. In the shell finite element model, the details of the connecting welds are not modeled, while the welds are modeled in detail in the solid finite element model. In order to make a direct comparison between the shell and the solid finite element models, these models must have the same dimensions, same material properties, and subjected to the same boundary conditions and loading.

Two steam dryer components are being analyzed in this calculation:

1. The cover plate, which is welded to the outer hood (Hot Spot 1 of Figure 1-1).

2. The outer hood, which is welded to the gusset (Hot Spot 2 of Figure 1-1).

Stress profiles from the full model analysis (Reference 3) are provided for each of the two components. Appropriate loading conditions that closely match the original stress profiles will be used. This ensures that the loading that causes the localized high stress is captured.

The finite element analyses in this calculation are performed using ANSYS Version 11.0 (References 5 and 6).

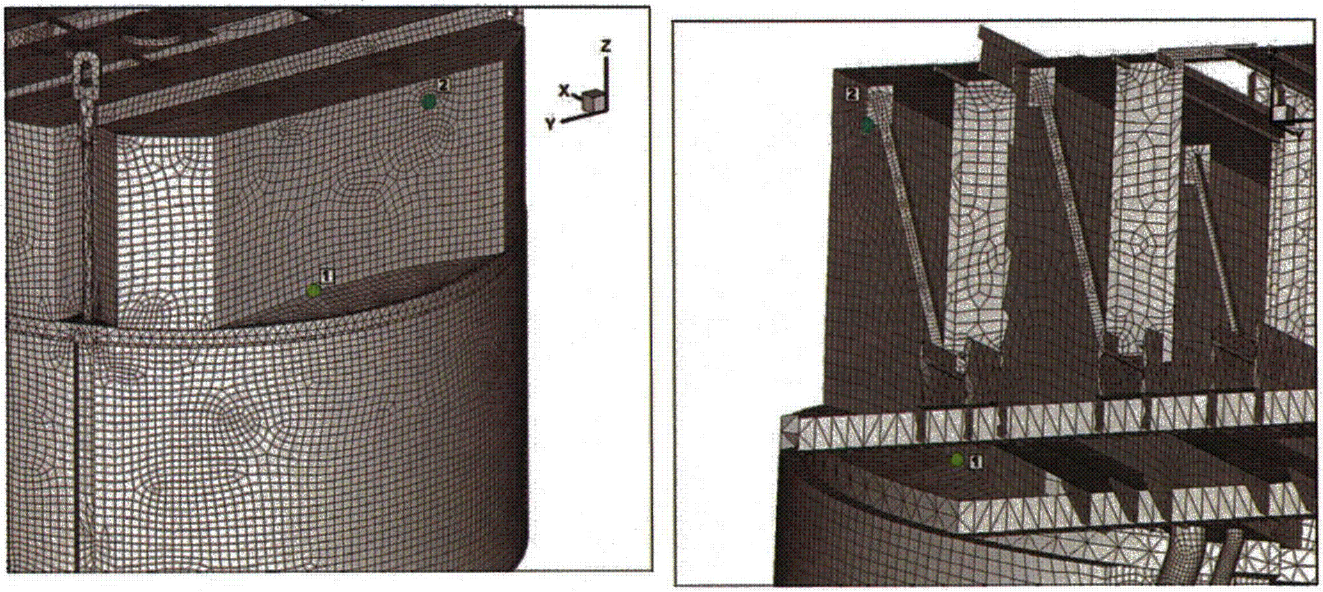


Figure 1-1: Hot Spots Locations

2.0 METHODOLOGY AND ASSUMPTIONS

Cover Plate

The full model analysis shows that the maximum stress is essentially bending stress (P_b) with little membrane stresses (P_m). When the linearized stresses are obtained for the different locations in the solid model, both the membrane stress (P_m) and the membrane plus bending stress ($P_m + P_b$) are computed.

The evaluation only compares the membrane plus bending stress ($P_m + P_b$) of the solid model with the membrane plus bending stress ($P_m + P_b$) of the shell model. The computed stress ratio will be used to adjust the stresses in the full model.

The stress profile for the full model shows that the highest stress is localized at the node that is the intersection at the outer hood and the cover plate at horizontal and slope sections. A unit load of 38.9 lb is applied to 28 outer hood nodes located about 7 inches above the cover plate.

Outer Hood

The full model analysis shows that the maximum stress is essentially bending stress (P_b) for the outer hood, and membrane stresses (P_m) for the connected gusset. When the linearized stresses are obtained for the different locations in the solid model, both the membrane stress (P_m) and the membrane plus bending stress ($P_m + P_b$) are computed.

The evaluation only compares the membrane plus bending stress ($P_m + P_b$) of the solid model with the membrane plus bending stress ($P_m + P_b$) of the shell model. The computed stress ratio will be used to adjust the stresses in the full model.

The stress profile for the full model shows that the highest stress is localized at the node located at the bottom of the outer hood and gusset connection. A load of 9,397.3 lb is applied to top plate and 28.9 lb is applied at the bottom end of the support brace.

Key Assumptions

1. A submodel is used for comparing the maximum stress intensity between the full shell and the shell submodel. In the submodels, appropriate boundary conditions are assumed.

2. The weld material is assumed to be the same as the base metal material.

3.0 MATERIAL PROPERTIES

The material properties used are as follows:

Modulus of Elasticity	=	25.55E6 psi	(Reference 4)
Poisson's Ratio	=	0.30	(Reference 4)

4.0 COVER PLATE ANALYSIS

The cover plate submodel consists of six key parts: the cover plate, the outer hood, the end panel, the top plate, the gusset, and the support brace. For the solid submodel, the weld is also included.

4.1 Key Dimensions

The key dimensions of the submodel are as follows:

Table 4-1: Cover Plate Submodel Key Dimensions

Component	Thickness / Size	Modeled Dimensions [2]
Cover Plate	1/4"	~14" (depth) x ~54 (width)
Outer Hood	1/2"	~54" (width) x ~62" (height)
Side Hood	1/2"	~29" (width) x ~62" (height)
Top Plate	1/2"	18.5" (depth) x ~75" (width)
Gusset	1/2"	5" (width) x 7" (height)
Support Brace	3/8"	2" x 2" x ~54" (length)
Cover Plate Weld [1]	1/4"	One sided fillet weld
Top Plate Weld [1]	1/2" / 3/16"	Both sided fillet weld
Gusset Weld [1]	1/4"	Both sided fillet weld
Support Brace Weld [1]	5/16"	Both sided fillet weld

Notes: (1) The fillet weld is modeled in the solid submodel.

(2) The dimensions are taken from the drawings (Reference 1).

4.2 Boundary Conditions and Applied Load

The boundary conditions and applied load are identified in Figure 4-1.

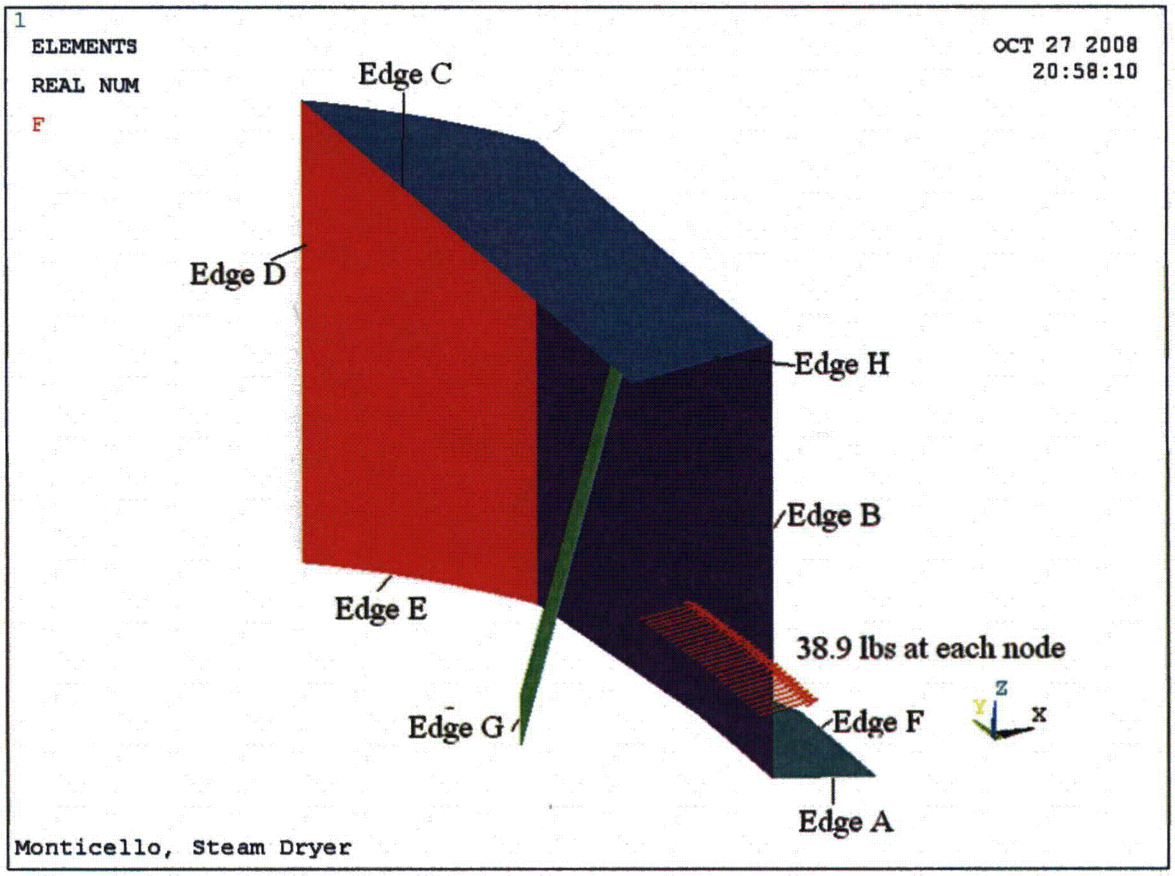


Figure 4-1: Cover Plate Model Boundary Conditions and Applied Load

<u>Boundary Conditions</u>

Edges A, B, C, D, H: Plane of symmetry.

Edges C, E, F: Restrained in Z translation.

Edge G: Restrained in all three directions.

<u>Applied Load</u>

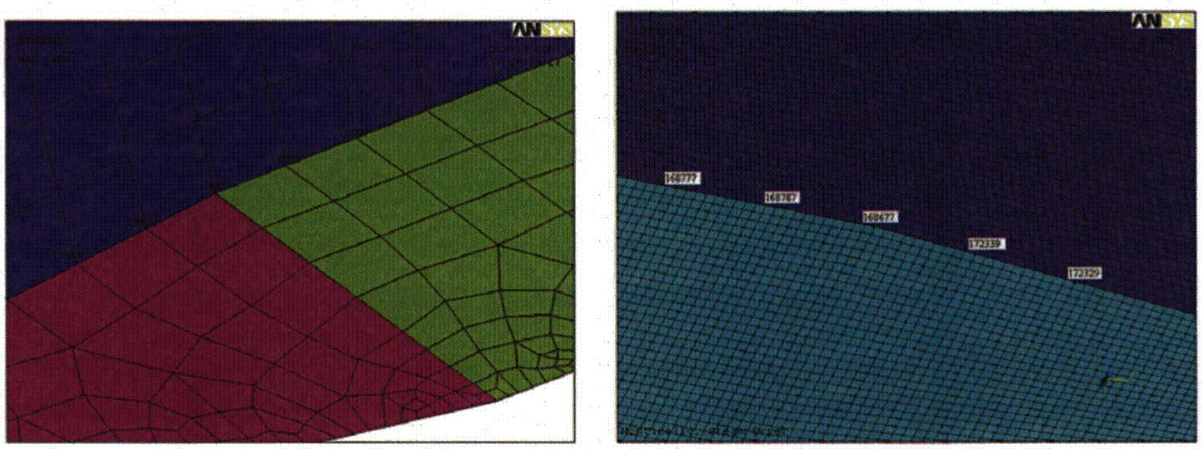
Load: 38.9 lb is applied at outer hood 28 nodes, located at about 7 inches above the cover plate.

The comparison of the stress profiles between the submodel and the full model is provided in Table 4-2 and Figure 4-2.

Table 4-2: Cover Plate Stress Profile Comparison

CDI Shell Model		Shell Submodel	
Node	SI (psi) [1]	Node	SI (psi)
139068	2,415	172329	2,965
139053	3,178	172339	3,164
139052	4,590	168677	4,590
139051	3,374	168787	3,361
139077	2,791	168777	3,218

Note: 1. These values are taken from References 3 and 2.



CDI Shell Model Shell Submodel

Figure 4-2: CDI Shell Model and Shell Submodel at the Cover Plate

4.3 Shell Finite Element Model

The shell finite element model is modeled using SHELL63 elements. A regular mesh size of 0.25" is used for the entire model. The entire model consists of approximately 97,000 nodes and 96,000 shell elements. The finite element mesh is shown in Figure 4-3.

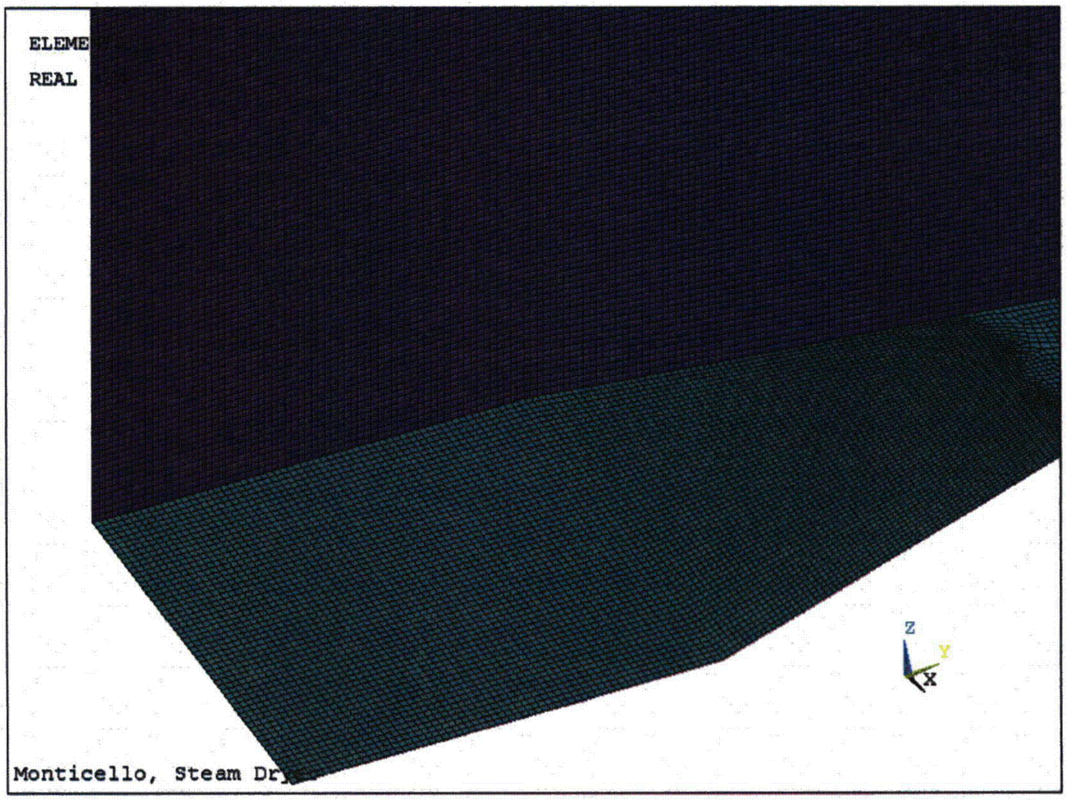


Figure 4-3: Cover Plate Shell Submodel Finite Element Model Mesh

4.4 Solid Finite Element Model

The solid finite element model is modeled using SOLID45 elements. A regular mesh size of 0.25" is used throughout the model, except in the transition regions. Six layers of element are modeled across the plate thickness, therefore, providing adequate discretization through the plate thickness to capture the stress variations across the thickness. The entire model consists of approximately 888,000 nodes and 759,000 solid elements. The finite element mesh is shown in Figure 4-4 and Figure 4-5.

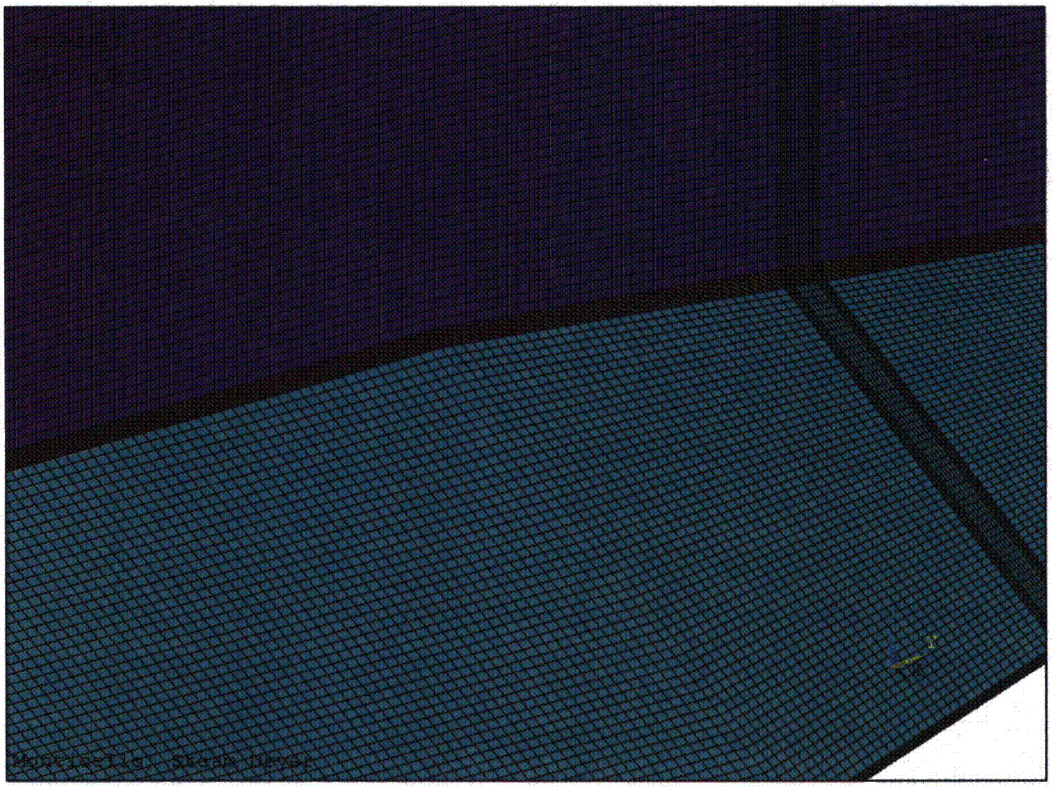


Figure 4-4: Cover Plate Solid Finite Element Model Mesh (Isometric View)

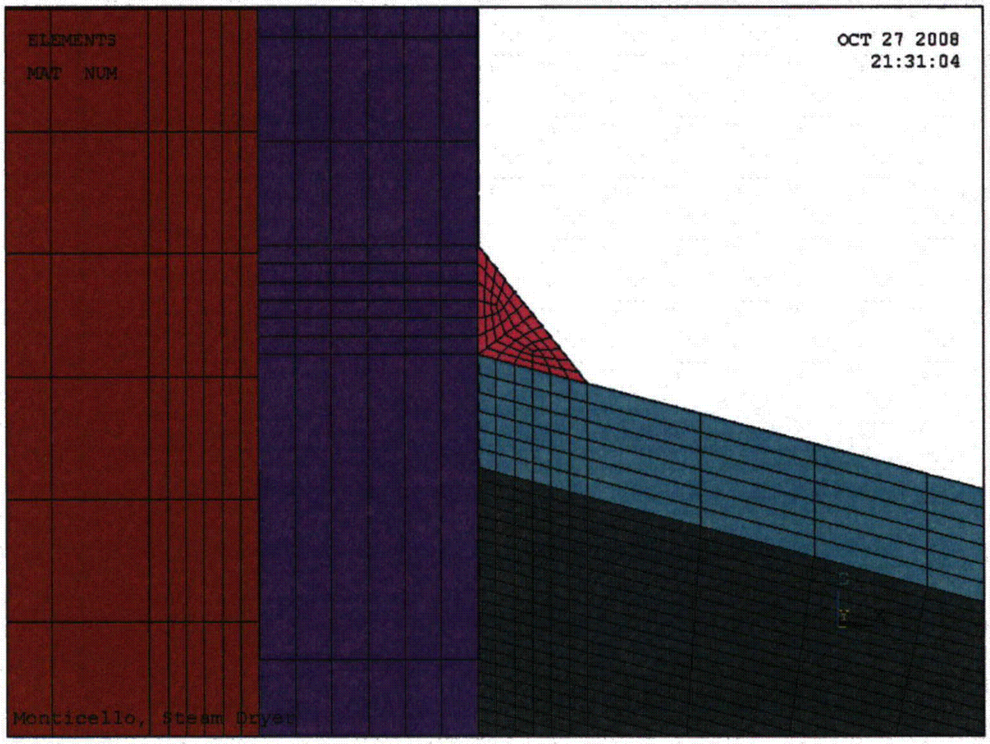


Figure 4-5: Cover Plate Solid Finite Element Model Mesh (Side View)

4.5 Solid Model Stress Paths

Linearization stress paths are taken from the weld root to the component surface in the vicinity of the high stress region. In addition, linearization stress paths are also taken from the weld toe to the opposite surface of the connected parts. The stress paths used for the Cover Plate solid model are shown in Figure 4-6.

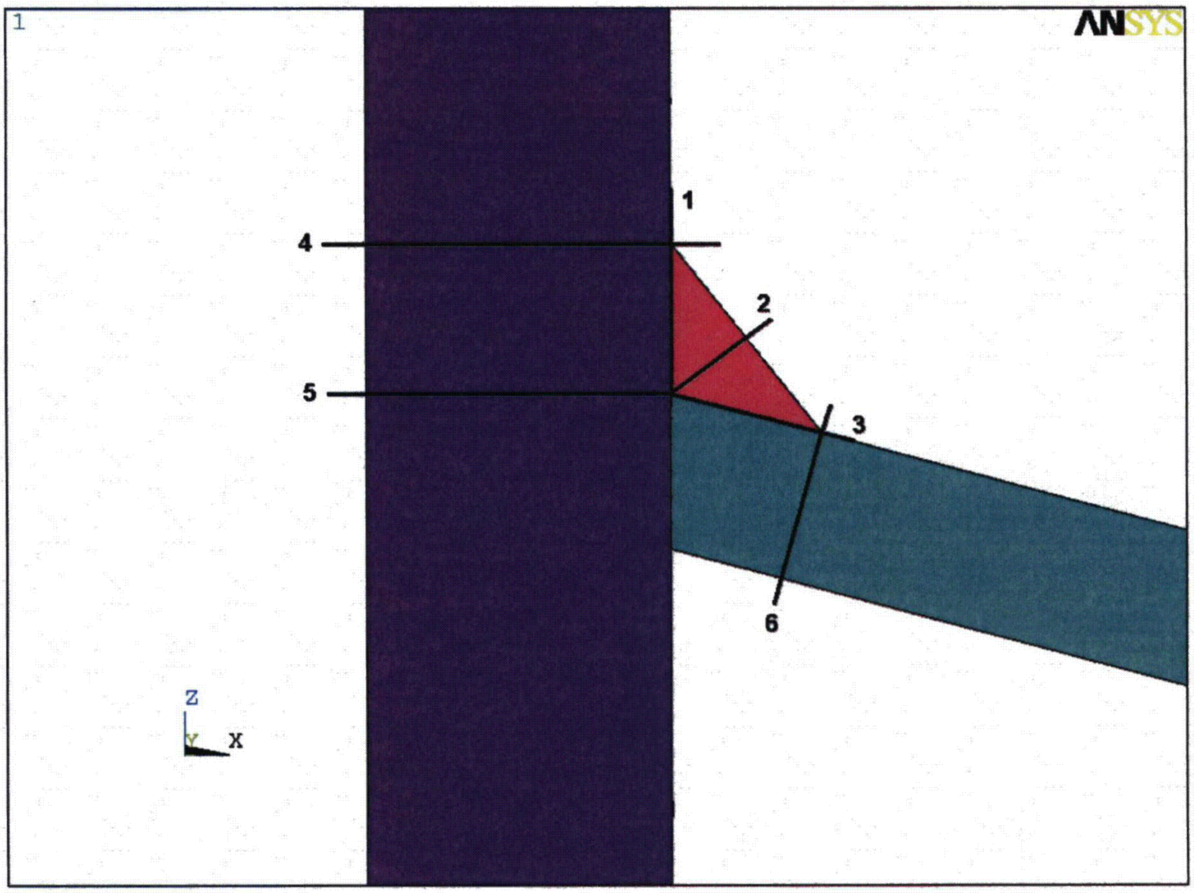


Figure 4-6: Cover Plate Solid Model Stress Paths

~~Contains Vendor Proprietary Information~~

4.6 Shell Model Results

The stress intensity plot for the shell model is provided in Figure 4-7.

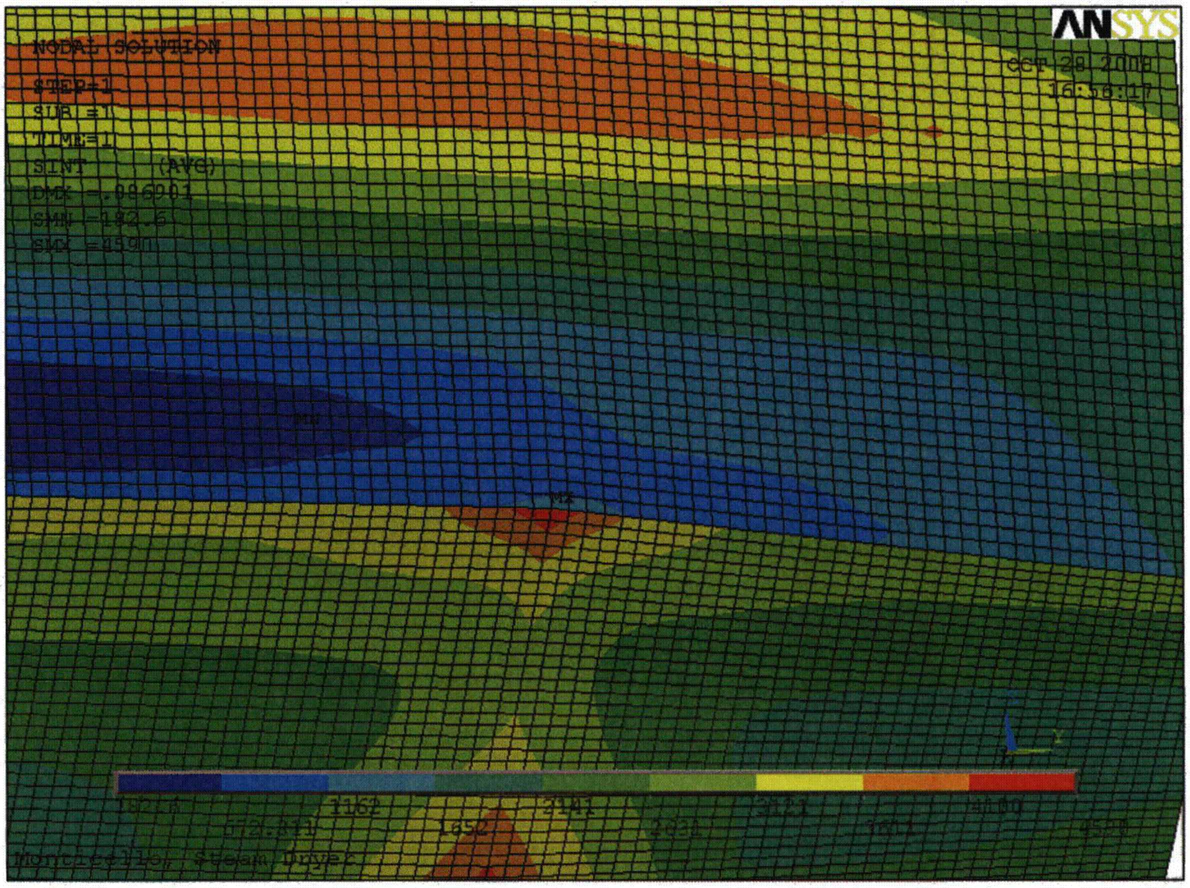


Figure 4-7: Cover Plate Shell Model Stress Intensity at High Stress Location

<u>Summary</u>

The maximum $P_m + P_b$ stress intensity is 4,590 psi.

4.7 Solid Model Results

The stress intensity plot for the solid model is provided in Figure 4-8.

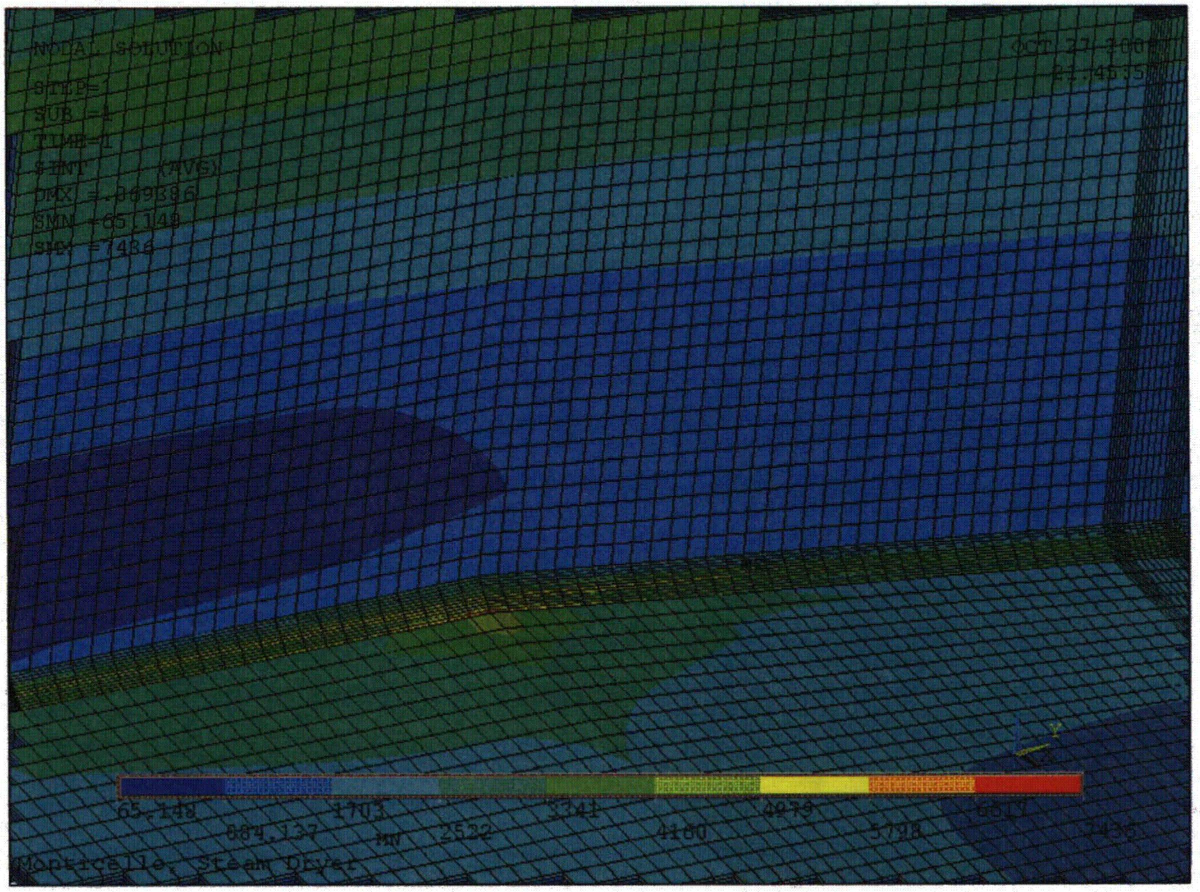


Figure 4-8: Cover Plate Solid Model Stress Intensity at High Stress Region

Maximum Stress Intensity

The stress intensity is computed for the different stress paths. There are multiple locations for each of the paths, and the largest magnitude is identified as the maximum stress intensity for that path. The maximum stress intensity for the different paths are summarized in Table 4-3.

Table 4-3: Cover Plate Solid Model Maximum Stress Intensity

Path #	Pm+Pb (psi)
1	2,259
2	4,158
3	4,447
4	1,492
5	1,438
6	3,615

4.8 Stress Comparison

The stress comparison between the shell and the solid finite element models is summarized in Table 4-4.

Table 4-4: Cover Plate Solid Model / Shell Model Stress Ratio

Path #	Solid (psi)	Shell (psi)	Stress Ratio
1	2,259	4,590	0.49
2	4,158		0.91
3	4,447		0.97
4	1,492		0.33
5	1,438		0.31
6	3,615		0.79
		Maximum =	**0.97**

<u>Summary</u>

The maximum stress ratio for the solid model stress intensity / shell model stress intensity is 0.97.

5.0 OUTER HOOD ANALYSIS

The outer hood submodel is the same as cover plate submodel that is described in Section 4.0.

5.1 Key Dimensions

The key dimensions of the submodel are provide in Table 4-1.

5.2 Boundary Conditions and Applied Load

The boundary conditions and applied load are identified in Figure 5-1.

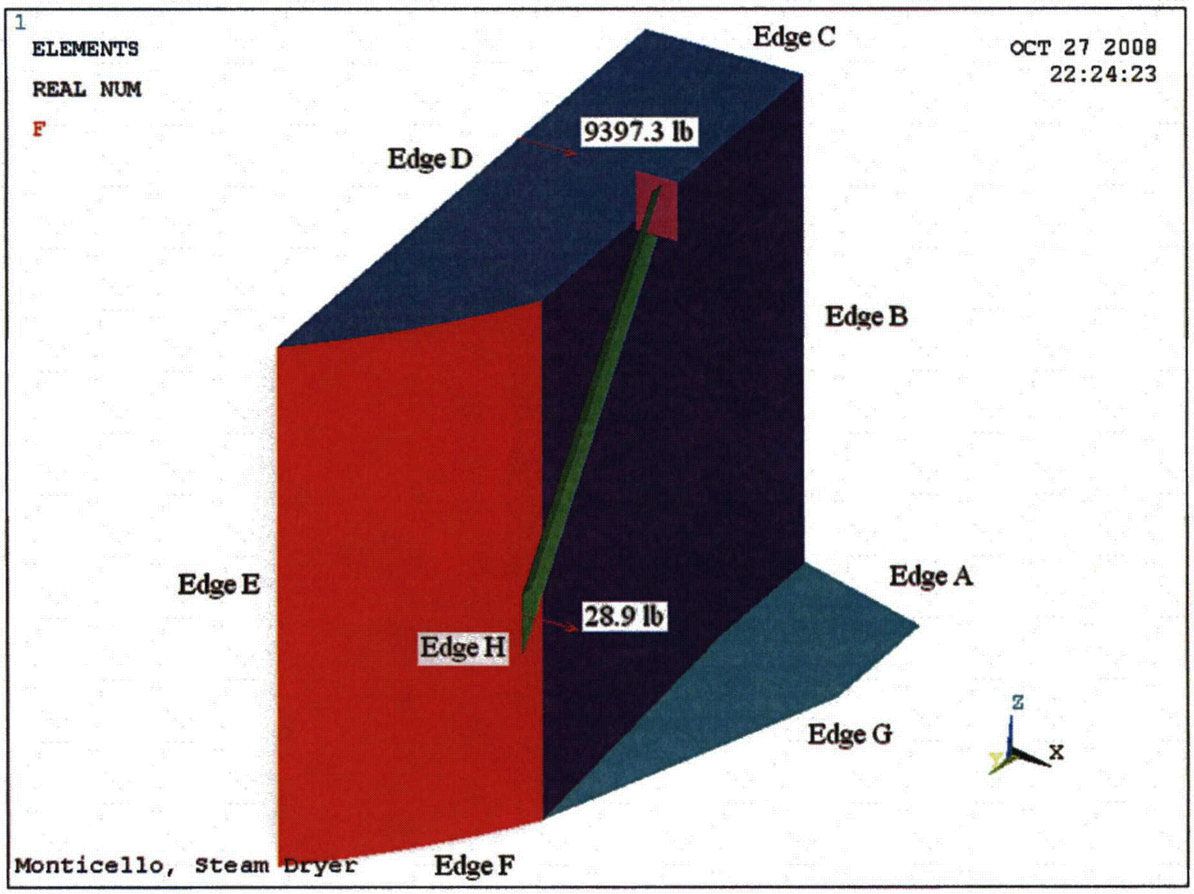


Figure 5-1: Outer Hood Model Boundary Conditions and Applied Load

<u>Boundary Conditions</u>

Edges A B, C, E: Plane of symmetry.

Edges F, G: Restrained in Z translation.

<u>Applied Load</u>

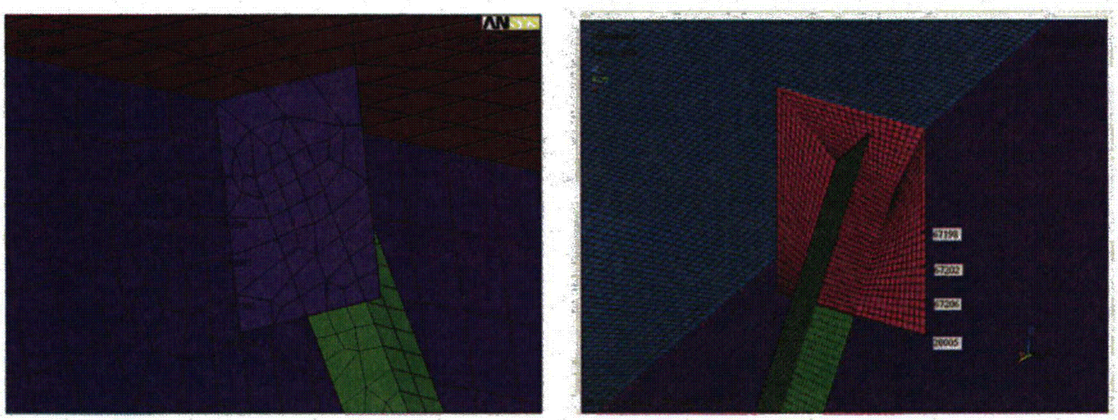
1. Apply a total load of 28.9 lb in the X direction at Edge H nodes.
2. A load of 9,397.3 lb is applied in the X direction along Edge D, at a node that corresponds to the gusset location in the Y direction.

The comparison of the stress profiles between the submodel and the full model is provided in Table 5-1 and Figure 5-2.

Table 5-1: Outer Hood Stress Profile Comparison

CDI Shell Model		Shell Submodel	
Node	SI (psi)[1]	Node	SI (psi)
140301	4,303	20005	5,769
140300	952	67206	3,622
140307	642	67202	3,561
140306	530	67198	3,768

Note: 1. These values are taken from Reference 3.



CDI Shell Model Sub-Shell Model

Figure 5-2: CDI Shell Model and Shell Submodel at the Outer Hood

5.3 Shell Finite Element Model

The shell finite element model is modeled using SHELL63 elements. A regular mesh size of 0.25" is used for the entire model. The entire model consists of approximately 99,000 nodes and 98,000 shell elements. The finite element mesh is shown in Figure 5-3.

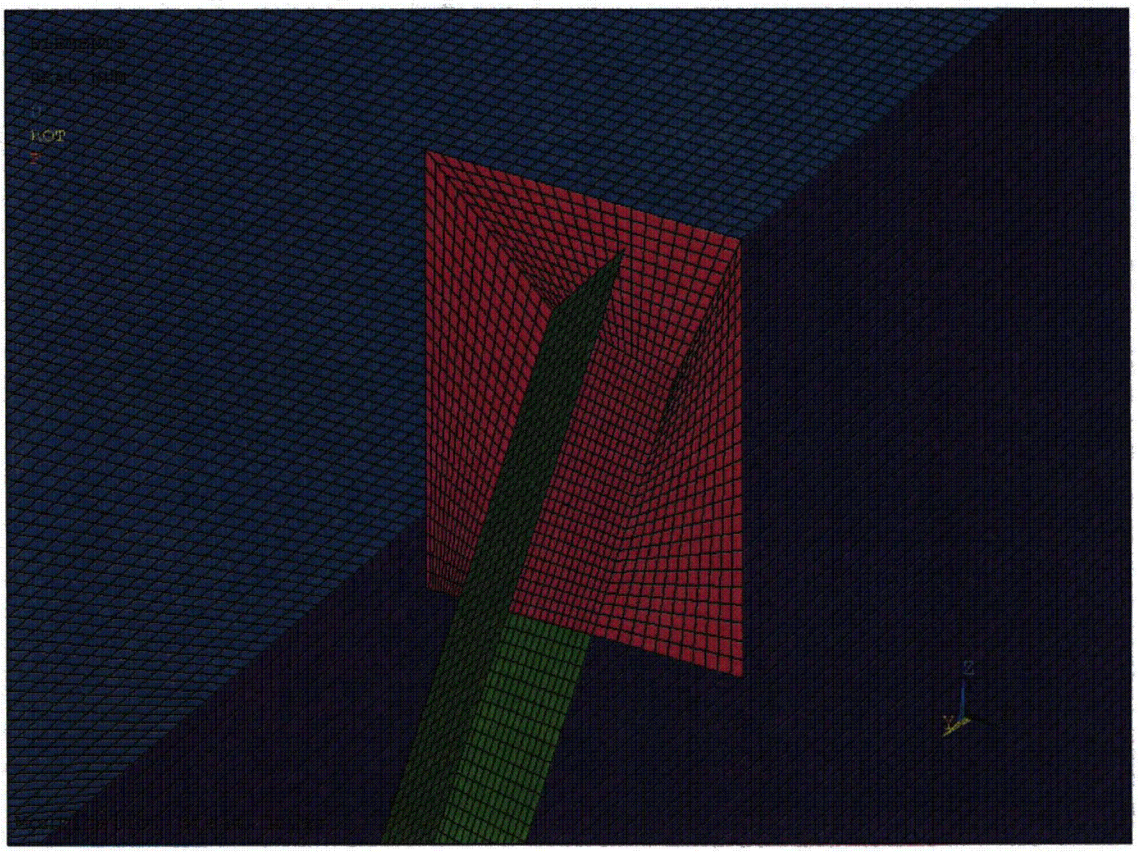


Figure 5-3: Outer Hood Shell Finite Element Model Mesh

5.4 Solid Finite Element Model

The solid finite element model is modeled using SOLID45 elements. A regular mesh size of 0.25" is used throughout the model, except in the transition regions. Six layers of element are modeled across the plate thickness, therefore, providing adequate discretization through the plate thickness to capture the plate bending behavior. The entire model consists of approximately 902,000 nodes and 772,000 solid elements. The finite element mesh is shown in Figure 5-4 and Figure 5-5.

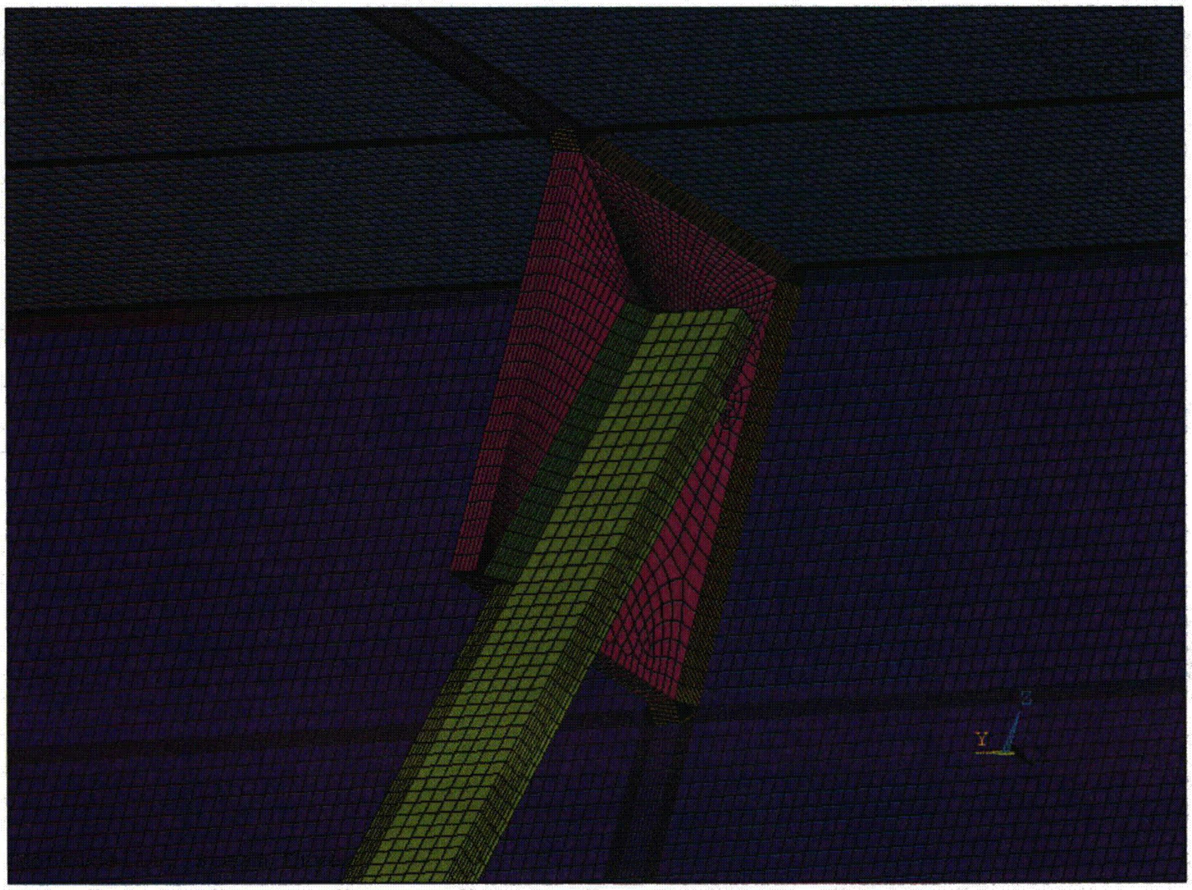


Figure 5-4: Outer Hood Solid Finite Element Model Mesh

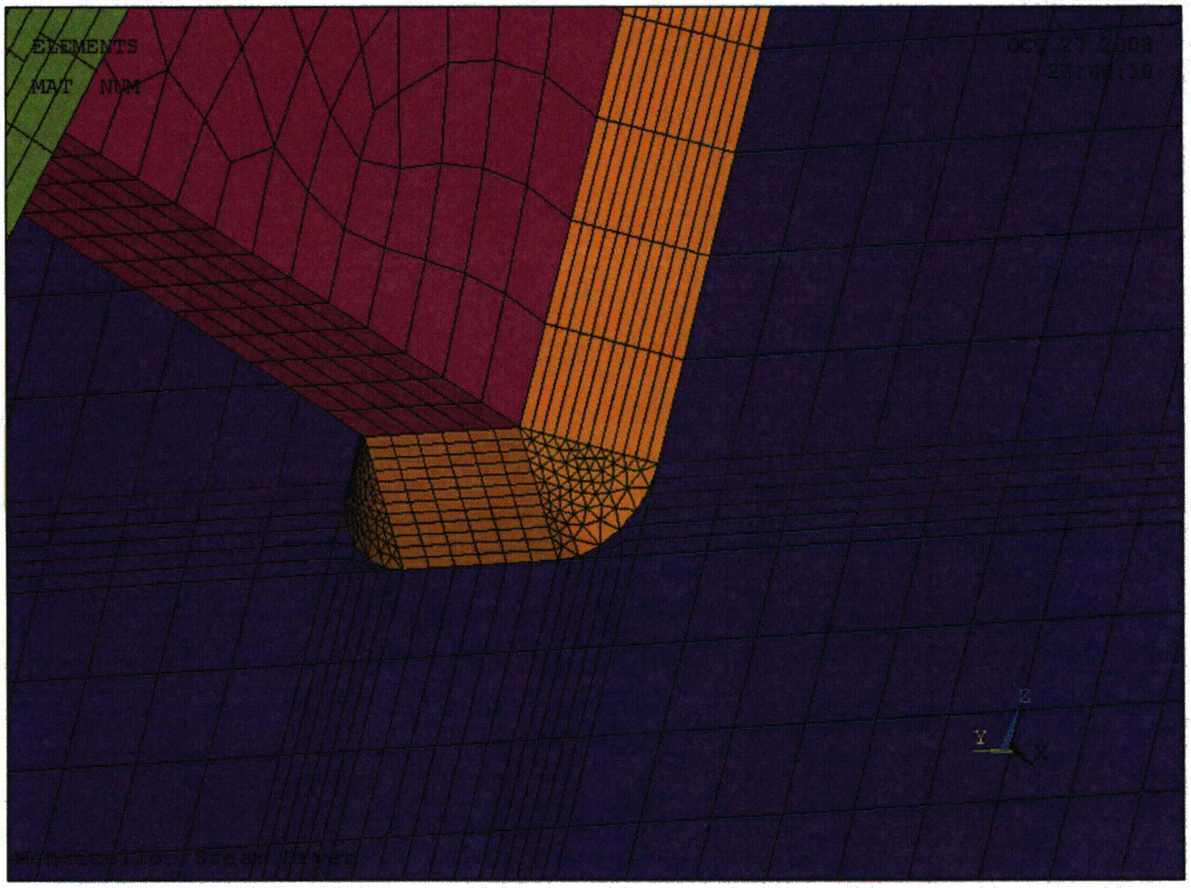


Figure 5-5: Outer Hood Solid Finite Element Model Mesh (Weld Region)

5.5 Solid Model Stress Paths

Linearization stress paths are taken from the weld root to the component surface in the vicinity of the high stress region. In addition, linearization stress paths are also taken from the weld toe to the opposite surface of the connected parts. The stress paths used for the outer hood solid model are shown in Figure 5-6 and Figure 5-7.

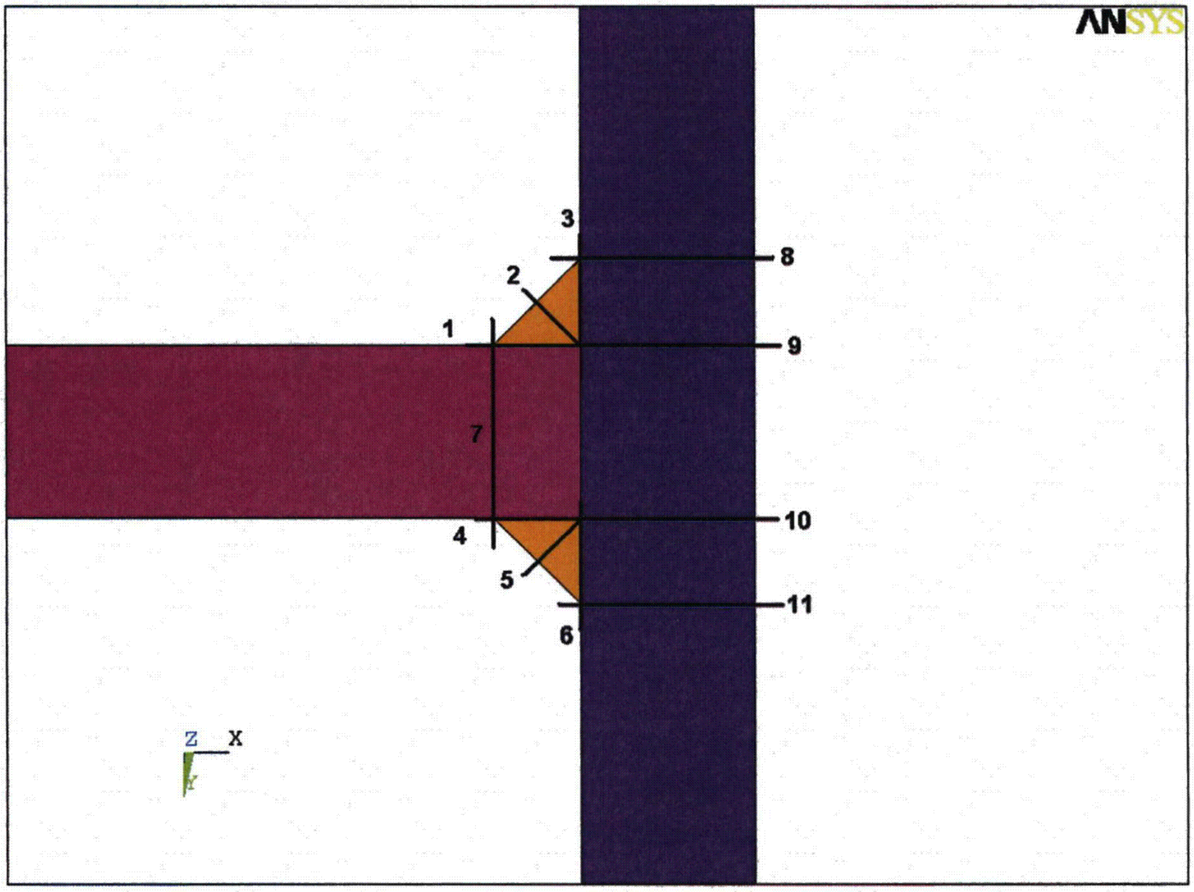


Figure 5-6: Outer Hood Solid Finite Element Stress Paths (End View)

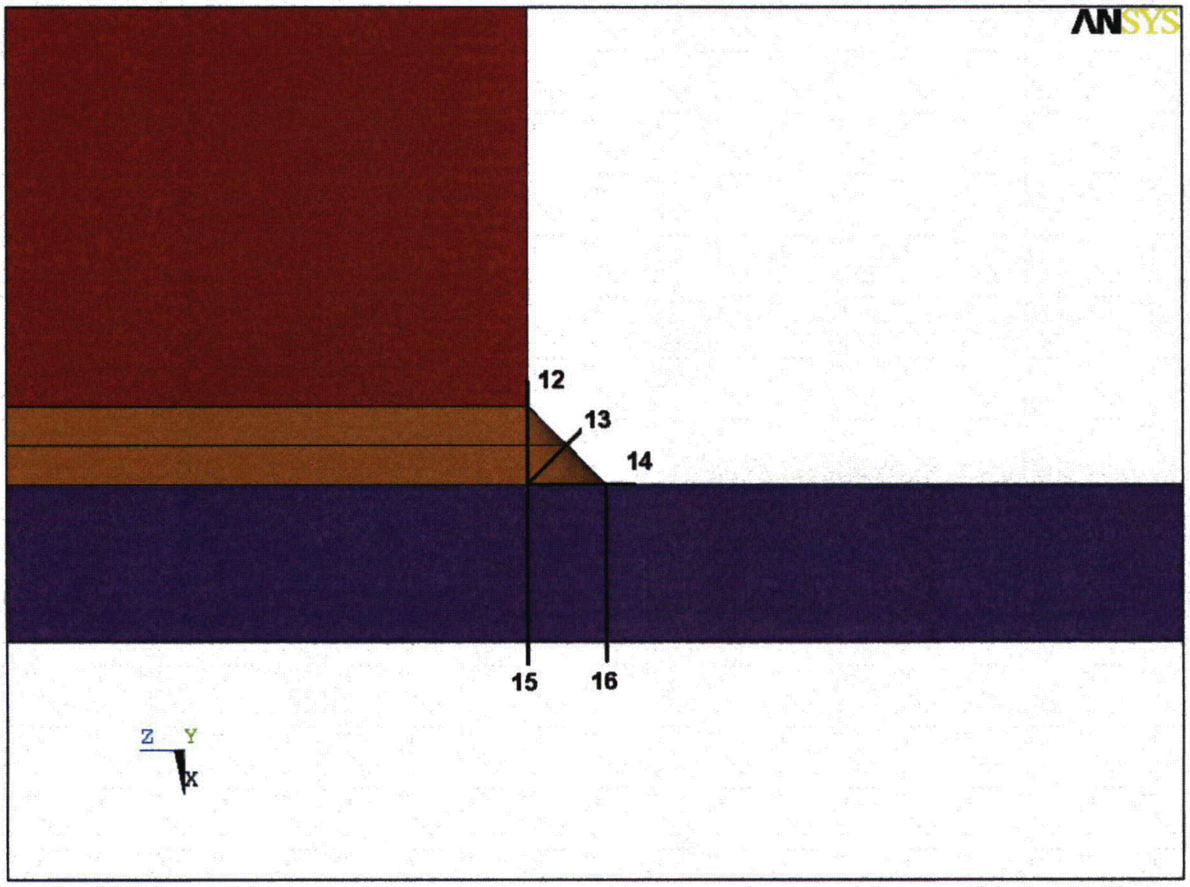


Figure 5-7: Outer Hood Solid Finite Element Stress Paths (Side View)

5.6 Shell Model Results

The stress intensity plot for the shell model at the bottom of the Outer Hood, which is the location of interest, is provided in Figure 5-8.

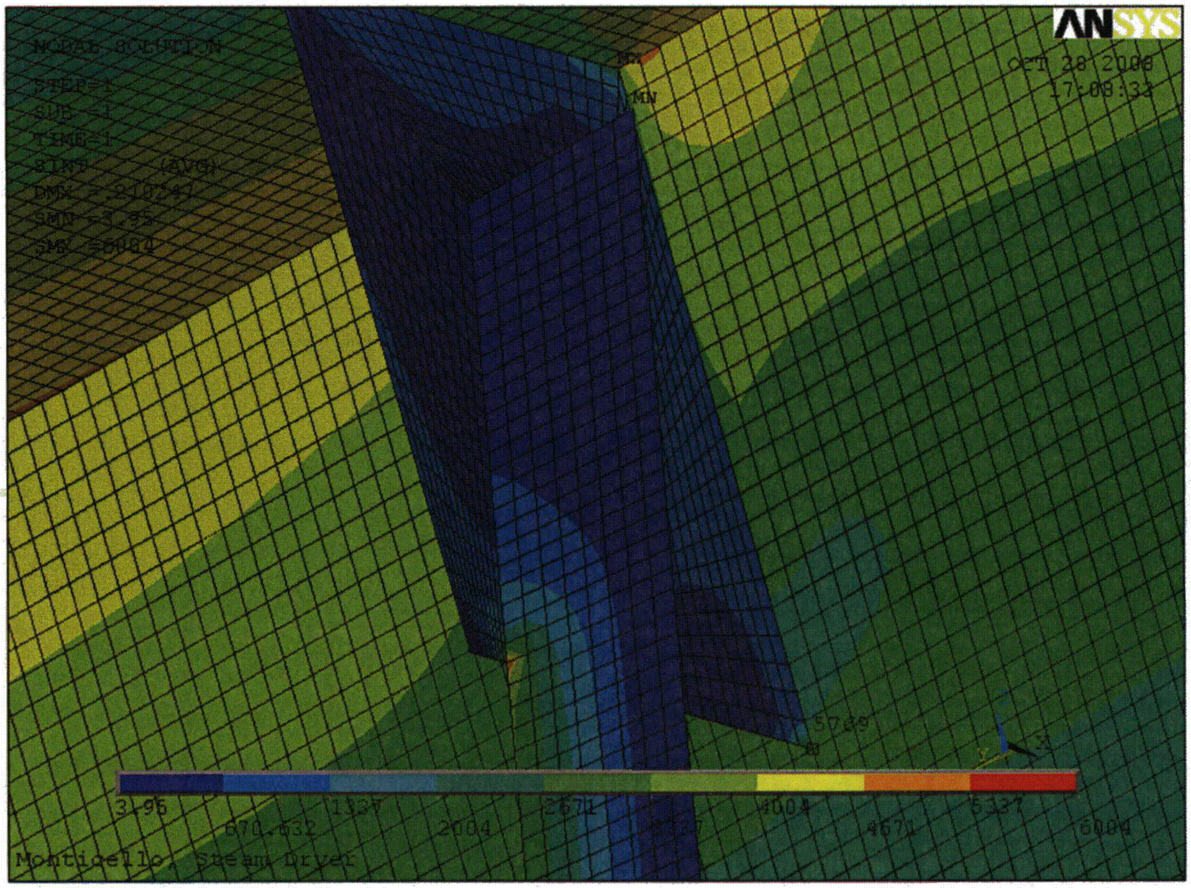


Figure 5-8: Outer Hood Shell Finite Element $P_m + P_b$ Stress Intensity

Summary

The maximum membrane plus bending stress ($P_m + P_b$) stress intensity at the bottom of the gusset is 5,769 psi.

5.7 Solid Model Results

The stress intensity plot for the solid model is provided in Figure 5-9.

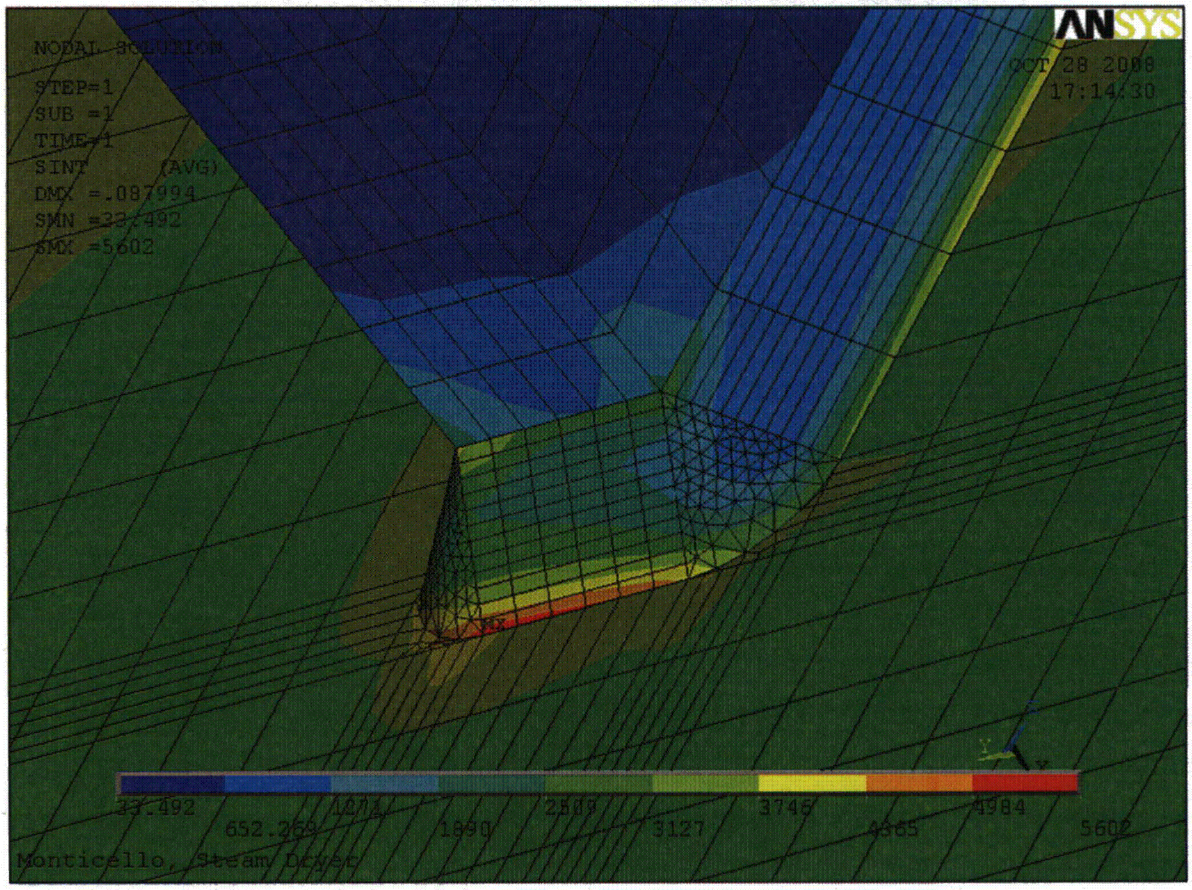


Figure 5-9: Outer Hood Solid Finite Element Stress Intensity at the Bottom of the Gusset (Inside Weld Region)

Maximum Stress Intensity

The stress intensity is computed for the different stress paths. There are multiple locations for each of the paths, and the largest magnitude is identified as the maximum stress intensity for that path. The maximum stress intensity for the different paths is summarized in Table 5-2.

Table 5-2: Outer Hood Solid Model Maximum Stress Intensity

Path #	Pm + Pb (psi)
1	1,665
2	933
3	1,096
4	1,313
5	835
6	1,017
7	1,443
8	882
9	966
10	966
11	878
12	1,665
13	1,129
14	1,395
15	773
16	985

5.8 Stress Comparison

The stress comparison between the shell and the solid finite element models is summarized in Table 5-3.

Table 5-3: Outer Hood Solid Model / Shell Model Stress Ratio

Path #	Solid (psi)	Shell (psi)	Stress Ratio
1	1,665	4,303	0.39
2	933		0.22
3	1,096		0.25
4	1,313		0.31
5	835		0.19
6	1,017		0.24
7	1,443		0.34
8	882		0.21
9	966		0.22
10	966		0.22
11	878		0.20
12	1,665		0.39
13	1,129		0.26
14	1,395		0.32
15	773		0.18
16	985		0.23
		Maximum =	**0.39**

<u>Summary</u>

The maximum stress ratio for the solid model stress intensity / shell model stress intensity is 0.39.

6.0 CONCLUSIONS

The stress intensity comparison shows that the stress intensities computed using the stress linearization approach for the solid elements (with the weld modeled) are lower than the stress intensities computed using the shell elements (without the weld modeled).

Cover plate

The solid model / shell model stress ratio for the cover plate is 0.97. This ratio is applicable to the cover plate shell stress intensity at the welded connection between the cover plate and the outer hood.

Outer hood

The solid model / shell model stress ratio for the outer hood is 0.39. This ratio is applicable to the outer hood shell stress intensity at the bottom of the welded connection between the outer hood and the gusset.

7.0 REFERENCES

1. Monticello Steam Dryer Drawings, SI File No. 0801040.202:
   a. General Electric Drawing, 729E913, "Steam Dryer."
   b. Stearns-Roger Drawing, 21775, Rev. 2, Sheet 3, "Final Field Assembly."
   c. Stearns-Roger Drawing, 21775, Rev. 4, Sheet 4, "Final Assembly."
   d. Stearns-Roger Drawing, 21775, Rev. 1, Sheet 6, "Shop Assembly Details."
2. Email with attachments from Alexander Boschitsch (CDI) to Soo Bee Kok (SI) on 10/21/08 at 3:53 am, "Re Additional Nodal Stress Needed," SI File No. 0801040.201P.
3. Email with attachment from Alexander Boschitsch (CDI) to Karen Fujikawa (SI) on 10/10/08 at 10:46 am, "Monticello Hot Spots," SI File No. 0801040.201P.
4. Email with attachment from Pavel Danilov (CDI) to Karen Fujikawa (SI) on 10/10/2008 at 10:34 am, "IBackup File – Folder Share – Monticello FEM," SI File No. 0801040.203.
5. ANSYS Mechanical, Release 11.0 (w/ Service Pack 1), ANSYS, Inc., August 2007.
6. SI calculation No. 0801040.303, Revision 0, "Project Specific Software Verification and Validation of ANSYS Release 11.0 (w/Service Pack 1)."

APPENDIX A

COMPUTER FILES

Filename	Description
SteamDryer_Shell-Top.inp	Cover Plate Shell Model Geometry input.
t11.inp	Cover Plate Shell Model Analysis input.
t11.oo	Cover Plate Shell Model load output.
SteamDryer_Solid-BOTTOM.inp	Cover Plate Solid Model Geometry input.
t22.inp	Cover Plate Solid Model Analysis input.
Pathbot.dat	Cover Plate Path Definition file.
PPpath2.mac	Post Process File for Extracting Stresses for Cover Plate.
t22.o	Cover Plate Solid Model Stress output.
t22pp.xls	Cover Plate Result Summary Spreadsheet.
Shelltop.inp	Outer Hood Shell Model Geometry input.
t77.inp	Outer Hood Shell Model Analysis input.
SteamDryer_Solid-TOP.inp	Outer Hood Solid Model Geometry input.
t44.inp	Outer Hood Solid Model Analysis input.
Pathtop.dat	Outer Hood Path Definition file.
t44.o	Outer Hood Solid Model Stress output.
t44pp.xls	Outer Hood Result Summary Spreadsheet.