APPENDIX 2.10.10

TABLE OF CONTENTS

Page

2.10.10 NUHOMS®-MP197 PACKAGE FINITE ELEMENT ANALSIS DETAILS

2.10.10,1 Introduction 2.10.10-1
2.10.10.2 3D Finite Element Model Description 2.10.10-2
2.10.10.3 Review of 3D FEM Warning Elements 2.10.10-14
2.10.10.4 Axisymmetric (2D) Finite Element Model Description 2.10.10-20
2.10.10.5 Review of 2D FEM Warning Elements 2.10.10-25
2.10.10.6 Maximum Component Stress Evaluation 2.10.10-26
2.10.10.7 Evaluation of Bearing Stresses in Flange 2.10.10-29
2.10.10.8 References 2.10.10-34

LIST OF TABLES

2.10.10-1 Individual Load Combinations
2.10.10-2 Normal Condition Load Combinations
2.10.10-3 Hypothetical Accident Load Combinations
2.10.10-4 Maximum Component Stresses for Normal Conditions
2.10.10-5 Maximum Component Stresses for Accident Conditions

LIST OF FIGURES

2.10.10-1	3D FEA of NUHOMS®-MP197 Transport Cask Showing the Components
2.10.10-2	Cask Outer Shell
2.10.10-3	Lead Shield
2.10.10-4	Cask Inner Shell
2.10.10-5	Contact Elements at Lead Shield / Shell Interface
2.10.10-6	CONTACT173 and TARGET170 Detail (Lead / Shell Interface) Near Top Flange
2.10.10-7	Cask Lid with Bolt Head Recess Detail
2.10.10-8	Shell Top Flange
2.10.10-9	Segment at Top Flange Assembly Showing Modeling Details – Lead Shield, Inner Shell, Lid, and Top Flange and Location of CONTACT52 Elements
2.10.10-10	Details of Bolt Assembly LINK8 Elements
2.10.10-11	Cask Bottom Cover and Flange
2.10.10-12	Cask Bottom Ram Cover Plate
2.10.10-13	Segment of Cask et the Bottom showing Assembly of Bottom Flange / Cover, Ram Cover Plate, Inner and Outer Shell.
2.10.10-14	Cask Bottom and Ram Cover Bolted Joints
2.10.10-15	Shear Key Bearing Block and Pad
2.10.10-16	Warning Elements from the 3D FEM
2.10.10-17	Warning Elements A from the 3D FEM in the Lid Near the Bolt Holes
2.10.10-18	Warning Group B from the 3D FEM in Top Flange near Bolt Holes
2.10.10-19	Location of Warning Group C Elements in the Cask Outer Shell
2.10.10-20	Warning Group D Elements in the Inner Shell Region
2.10.10-21	Warning Group E Elements in Bottom Plate / Flange and Ram Cover
2.10.10-22	Axisymmetric Finite Element Model of NUHOMS®-MP197 Transport Cask
2.10.10-23	Region 1, Showing Modeling Detail and Bolt Idealization
2.10.10-24	Details of Region 2
2.10.10-25	Region 3 Showing Element Idealization near the Bottom Flange
2.10.10-26	Region 4 Showing Ram Cover, Bottom Flange Detail
2.10.10-27	Warning Elements (Region 1) in Top Flange Area
2.10.10-28	Lid Finite Element Plot
2.10.10-29	Containment Body Finite Element Plot
2.10.10-30	Outer Shell Finite Element Plot
2.10.10-31	Run 13, Hot Environment (100° F) Stress Intensity Plot
2.10.10-32	Run 14, Cold Environment (-40° F) Stress Intensity Plot
2.10.10-33	Run 15, Increased External Pressure Stress Intensity Plot
2.10.10-34	Run 16, Minimum External Pressure Stress Intensity Plot
2.10.10-35	Run 17, Rail Car Vibration, Hot Environment Stress Intensity Plot
2.10.10-36	Run 18, Rail Car Vibration, Cold Environment Stress Intensity Plot
2.10.10-37	Run 19, Rail Car Shock, Hot Environment Stress Intensity Plot
2.10.10-38	Run 20, Rail Car Shock, Cold Environment Stress Intensity Plot

LIST OF FIGURES
(continued)

2.10.10-39	Run 21, 1 Foot Lid End Drop, Hot Environment Stress Intensity Plot
2.10.10-40	Run 22, 1 Foot Lid End Drop, Cold Environment Stress Intensity Plot
2.10.10-41	Run 23, 1 Foot Bottom End Drop, Hot Environment Stress Intensity Plot
2.10.10-42	Run 24, 1 Foot Bottom End Drop, Cold Environment Stress Intensity Plot
2.10.10-43	Run 25, 1 Foot Side Drop, Hot Environment Stress Intensity Plot
2.10.10-44	Run 26, 1 Foot Side Drop, Cold Environment Stress Intensity Plot
2.10.10-45	Run 27, 30 Foot Bottom End Drop, Hot Environment Stress Intensity Plot
2.10.10-46	Run 28, 30 Foot Bottom End Drop, Cold Environment Stress Intensity Plot
2.10.10-47	Run 29, 30 Foot Lid End Drop, Hot Environment Stress Intensity Plot
2.10.10-48	Run 30, 30 Foot Lid End Drop, Cold Environment Stress Intensity Plot
2.10.10-49	Run 31, 30 Foot Side Drop, Hot Environment Stress Intensity Plot
2.10.10-50	Run 32, 30 Foot Side Drop, Cold Environment Stress Intensity Plot
2.10.10-51	Run 33, 30 Foot CG over Bottom Corner Drop, Hot Environment Stress Intensity Plot
2.10.10-52	Run 34, 30 Foot CG over Bottom Corner Drop, Cold Environment Stress Intensity Plot
2.10.10-53	Run 35, 30 Foot CG over Lid Corner Drop, Hot Environment Stress Intensity Plot
2.10.10-54	Run 36, 30 Foot CG over Lid Corner Drop, Cold Environment Stress Intensity Plot
2.10.10-55	Run 37, 30 Foot 20° Lid End Slap Down Drop, Hot Environment Stress Intensity Plot
2.10.10-56	Run 38, 30 Foot 20° Lid End Slap Down Drop, Cold Environment Stress Intensity Plot
2.10.10-57	Run 39, 30 Foot 20° Bottom End Slap Down Drop, Hot Environment Stress Intensity Plot
2.10.10-58	Run 40, 30 Foot 20° Bottom End Slap Down Drop, Cold Environment Stress Intensity Plot
2.10.10-59	Run 41, Immersion (290 psi), Hot Environment Stress Intensity Plot
2.10.10-60	Run 42, Fire Accident, Cold Environment Stress Intensity Plot

This page intentionally blank

APPENDIX 2.10.10

NUHOMS®-MP197 PACKAGE FINITE ELEMENT ANALYSIS DETAILS

2.10.10.1 Introduction

This appendix provides additional detail information of the NUHOMS®-MP197 Transport Cask structural analysis. The primary focus of this appendix is the following.

- A detailed description of each component of the transport cask finite element model, including method of construction, mesh density, element types and evaluation of ANSYS warnings.
- An evaluation of the bearing load on the top surface of the flange where it contacts the bottom surface of the lid.
- An evaluation of the maximum stresses in each major component of the transport cask as well as the overall stress distribution.

2.10.10.2 3D Finite Element Model Description

The 3D finite element model of the NUHOMS®-MP197 Transport Cask is shown below in Figure 2.10.10-1. This model is symmetric about a diametrical plane. The details of the components are described below.

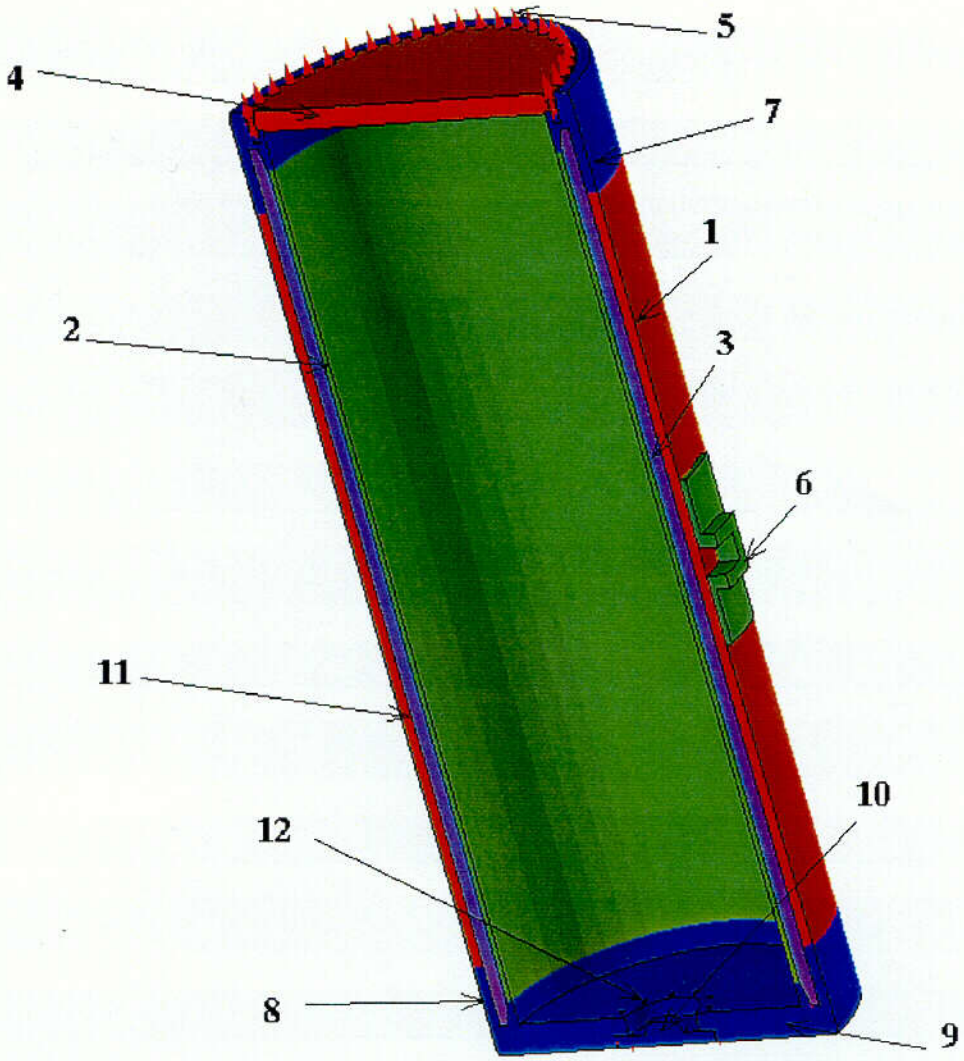


Figure 2.10.10-1
3D FEA model of NUHOMS®-MP197 Transport Cask Showing the Components.

1.Cask outer shell
2.Cask inner shell
3.Lead shield
4.Lid
5.Lid bolts
6.Shear key bearing block / Pad
7. Top flange region
8. Bottom flange region
9. Bottom
10. Ram cover plate
11. Lead /cask shell interface
12. Ram cover bolts

C01

The model consists of a total of 39,044 elements and 40,494 nodes. The assembly consists of the following elements from the ANSYS element library: SOLID45, CONTACT52, LINK8, TARGET170 and CONTACT173. A maximum aspect ratio of about 6.11 was observed in the model. An element error/warning check for the entire model invoked ANSYS warning messages (due to element aspect ratio and distortion (Jacobean errors)), that are summarized later. Sweeping and copying about the axis of the cask generated most of the elements. A brief description of the individual components is included. Circumferential shifts of some nodes have been made to accommodate the shear key bearing block and the trunnions. In regions where stress variations were significant, a finer mesh was generated relative to regions where only displacements and load transfer was of interest.

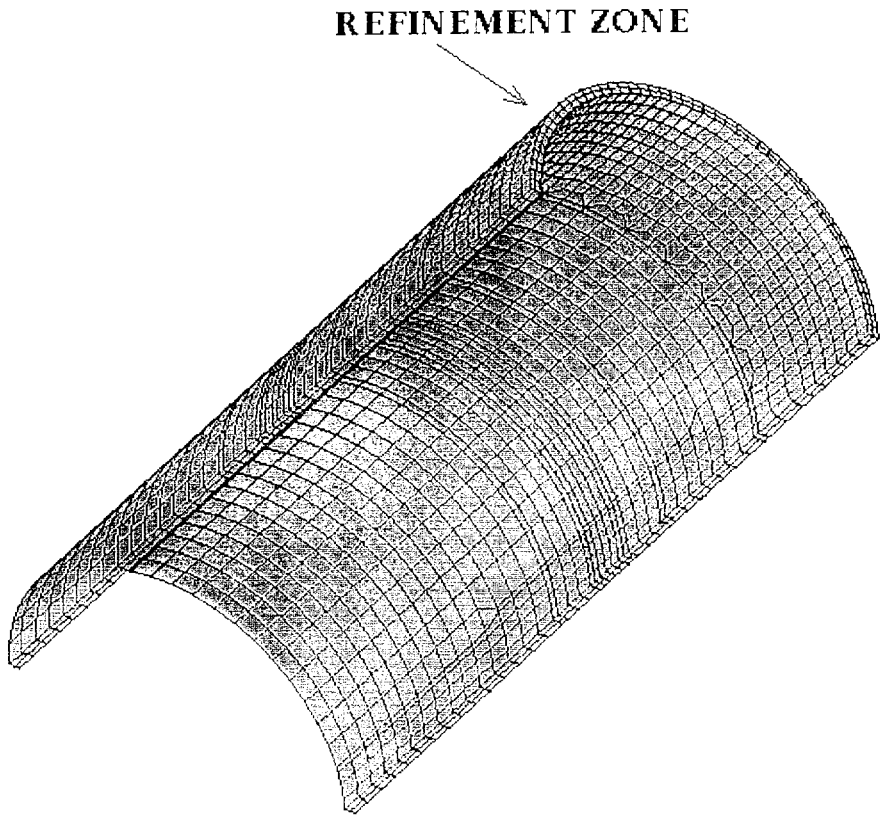


Figure 2.10.10-2: Cask Outer Shell (2,884 elements and 4,581 nodes)

The cask outer shell (Figure 2.10.10-2) is made of ANSYS SOLID45 elements. The maximum aspect ratio is 6.11. The elements were constructed using a combination of sweeping and copying about the cask centerline. The mesh is refined towards the lid to accommodate the details of the lid and the bolt assembly.

The lead shield is shown in Figure 2.10.10-3. This component is modeled with SOLID45 elements. There are, for the most part, two elements through the thickness with four elements through the thickness at the refinement zone to accommodate the details towards the top flange region. This part of the cask is isolated from the outer shell, inner shell, top and bottom flanges by surface-to-surface contact elements TARGET170 and CONTACT173. These elements provide contact and friction and allow relative movement between the lead shield and the shell. The elements are generated in a manner similar to the shell.

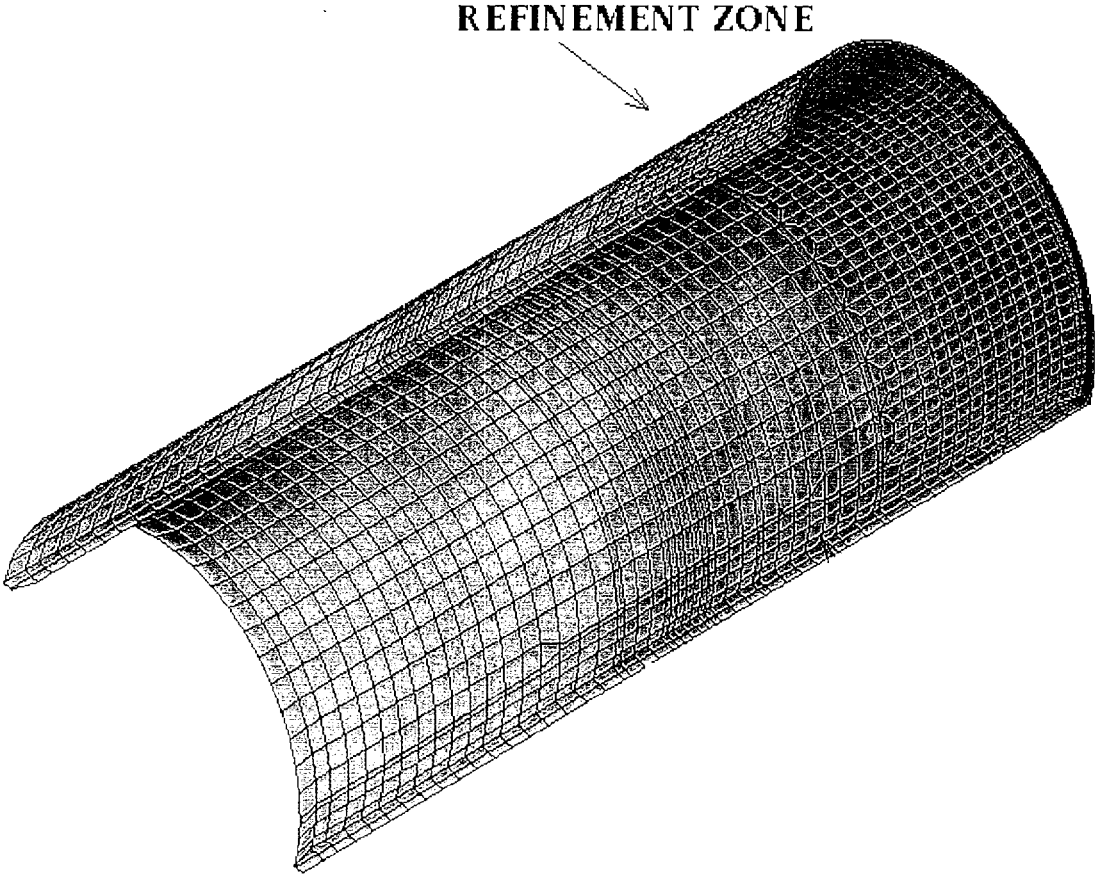


Figure 2.10.10-3: Lead Shield (4432 elements and 6588 nodes)

The FE mesh of the inner shell is shown in figure 2.10.10-4. Construction of the inner shell is similar to the outer shell with the necessary refinement towards the lid to accommodate the bolt assembly. The idealization is done using the SOLID45 element from the ANSYS element library. The maximum aspect ratio is about 6.0. The inner shell is separated from the lead lining by surface TARGET170 and CONTACT173. The ends of the shell connect to the top and bottom flanges. Symmetric boundary conditions are applied to the longitudinal edges for identical loading about the plane of symmetry.

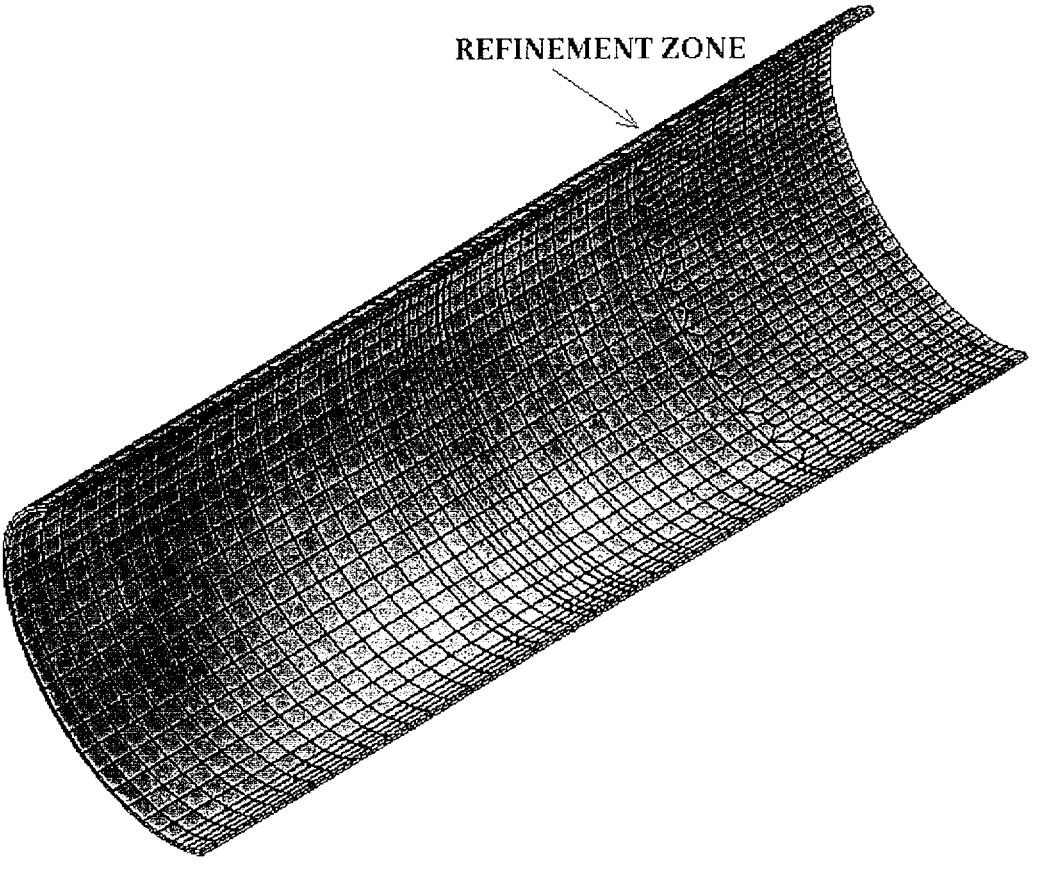
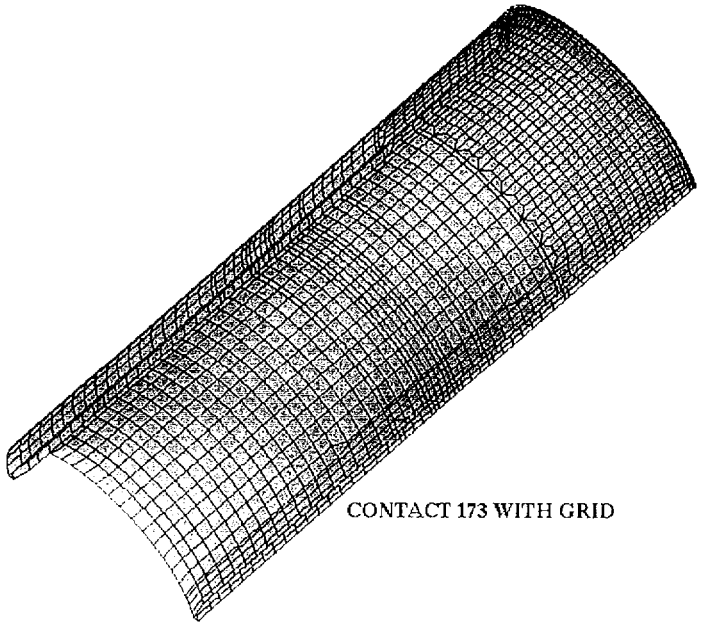


Figure 2.10.10-4: Cask Inner Shell (3276 elements and 5184 nodes)

Figures 2.10.10-5 and 2.10.10-6 show the details of the surface contact elements used. TARGET 170 and CONTACT173 are used together to establish contact and provide a friction surface to enable both surfaces to slide relative to each other. Figure 2.10.10-5 shows CONTACT173 with and without the gridlines. It can be seen that the surface configuration of the grid lines conforms to that of the lead liner. These contact elements envelop the lead liner completely. Construction of the TARGET170 contact is similar to CONTACT173 but conforms to the inner and outer shell along with the top and bottom flange assemblies. Figure 2.10.10-6 shows the grid differences between the two sets of contact elements at the end of the lead liner near the top flange.



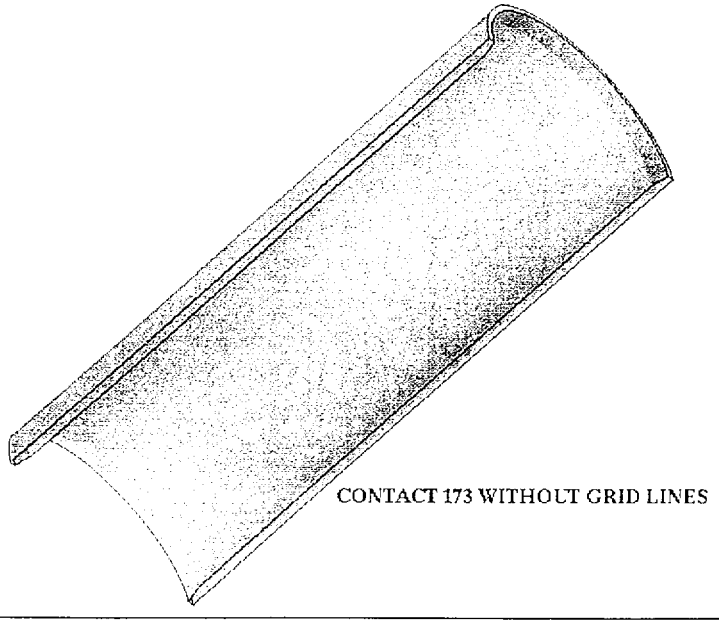


Figure 2.10.10-5: CONTACT173 Elements at Lead Shield / Shell Interface)

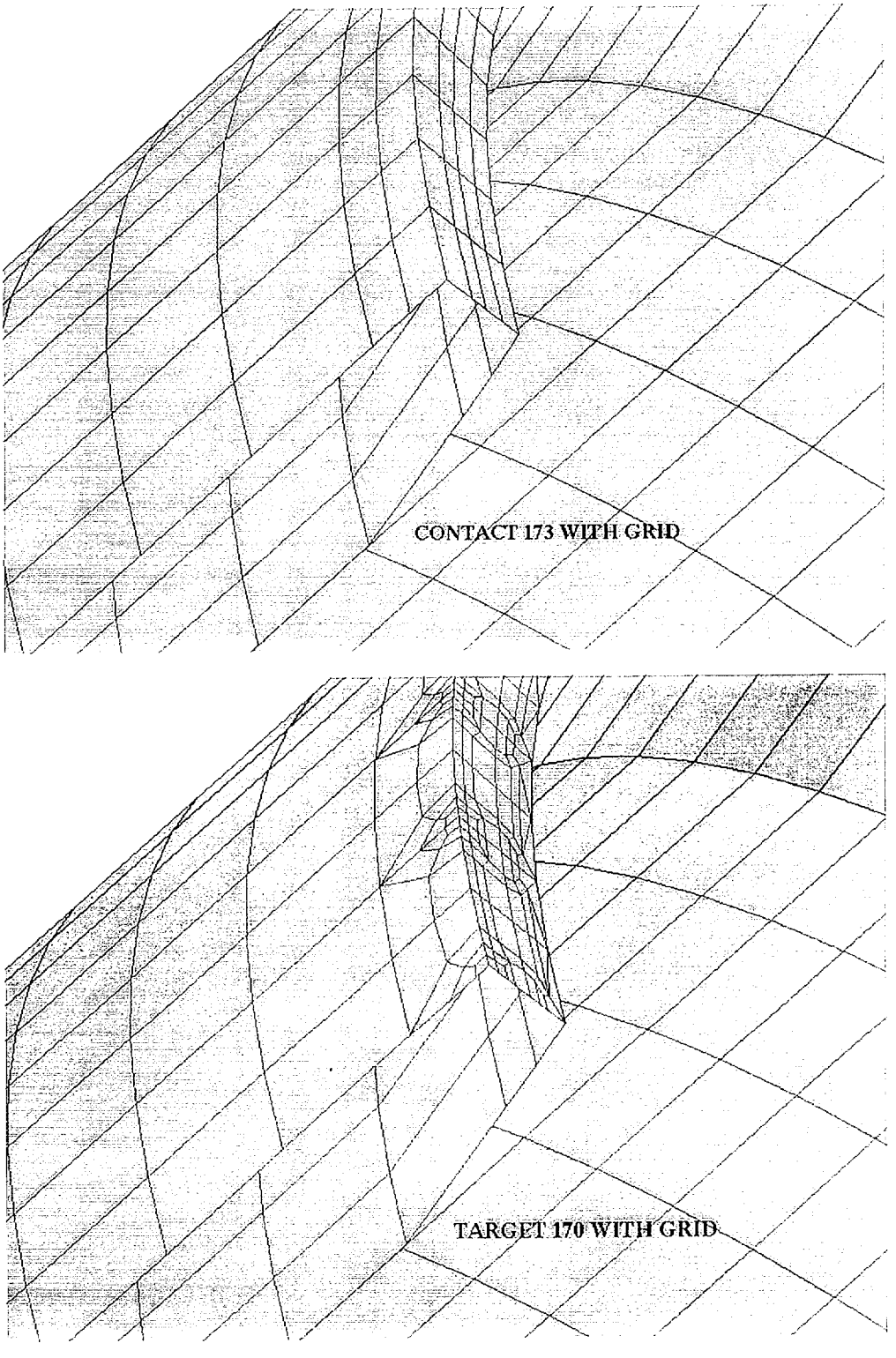


Figure 2.10.10-6: CONTACT173 and TARGET170 Detail (Lead / Shell Interface) Near the Top Flange.

Figure 2.10.10-7 shows the finite element idealization of the lid. Forty-eight bolts (twenty-four in the half model) connect the lid to the top shell flange. The lid is modeled with SOLID45 elements. The lid is connected to the top flange shell assembly (Figure 2.10.10-8) through ANSYS LINK8 clusters and CONTACT52 elements. The mesh has sufficient density to calculate the bending and extensional stresses accurately. The detail in Figure 2.10.10-7 shows the modeling details at the bolt recess.

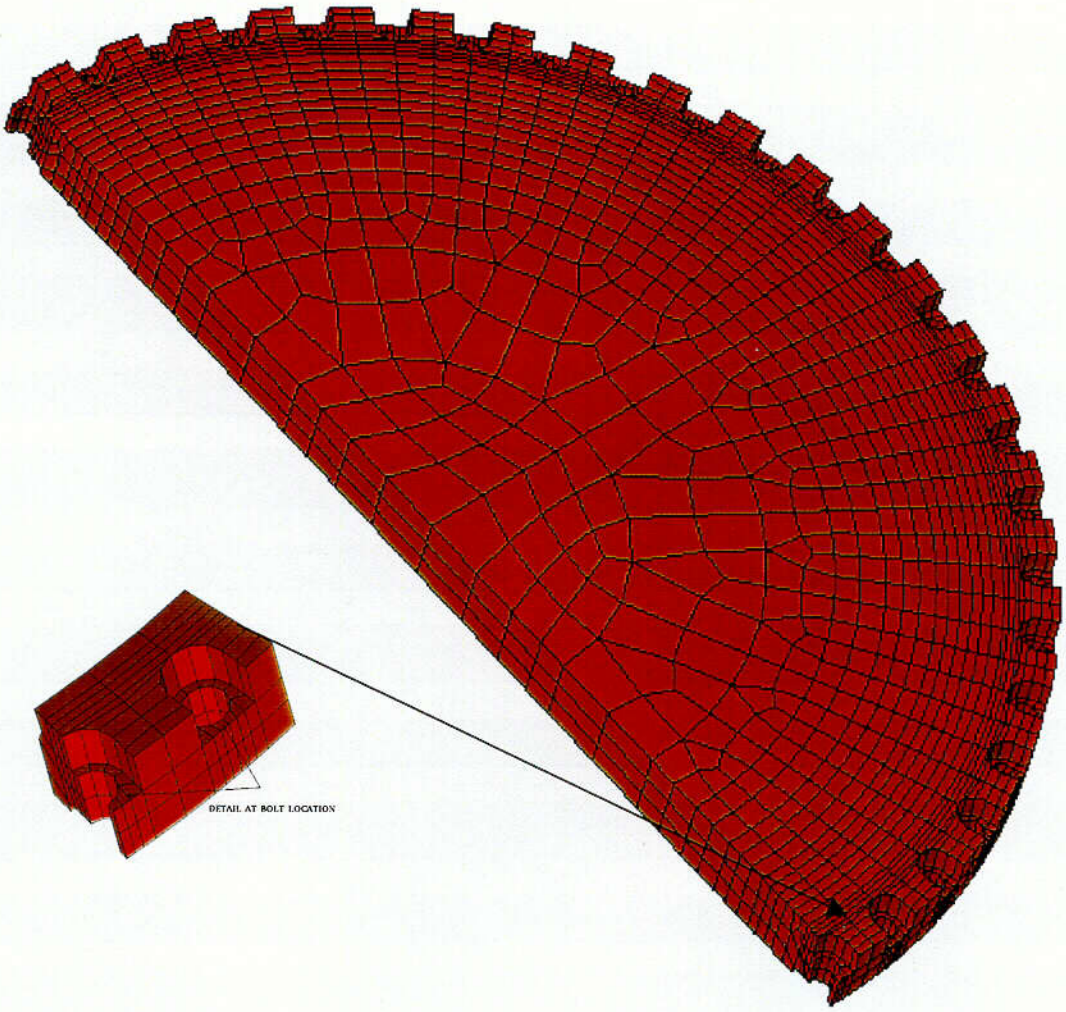


Figure 2.10.10-7: Cask Lid with Bolt Head Recess Detail (3828 Elements and 6124 Nodes)

The top flange and detail are shown in Figure 2.10.10-8. Figure 2.10.10-9 shows a segment of the assembly with details of modeling.

Figure 2.10.10-8: Shell Top Flange

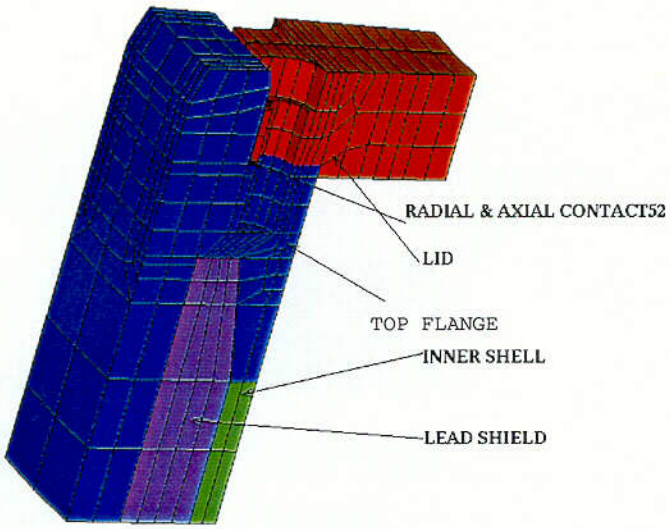


Figure 2.10.10-9: Segment at Top Flange Assembly Showing Modeling Details-Lead Shield, Inner Shell, Lid and Top Flange and Location of CONTACT52 Elements.

The details of the bolted joints between the top flange and the lid are shown in Figure 2.10.10-10. There are 48 bolted joints (24 in the half model) modeled with LINK8 elements. These are in addition to the CONTACT52 elements used in the radial and axial directions. A representative bolted joint consists of a shank representing the bolt (in area and material) and two sets of spider formations (LINK8 clusters) to transfer the loads. One set (green) connects to the counter bore and the other set (brown) to the bolt threads in the cask top flange.

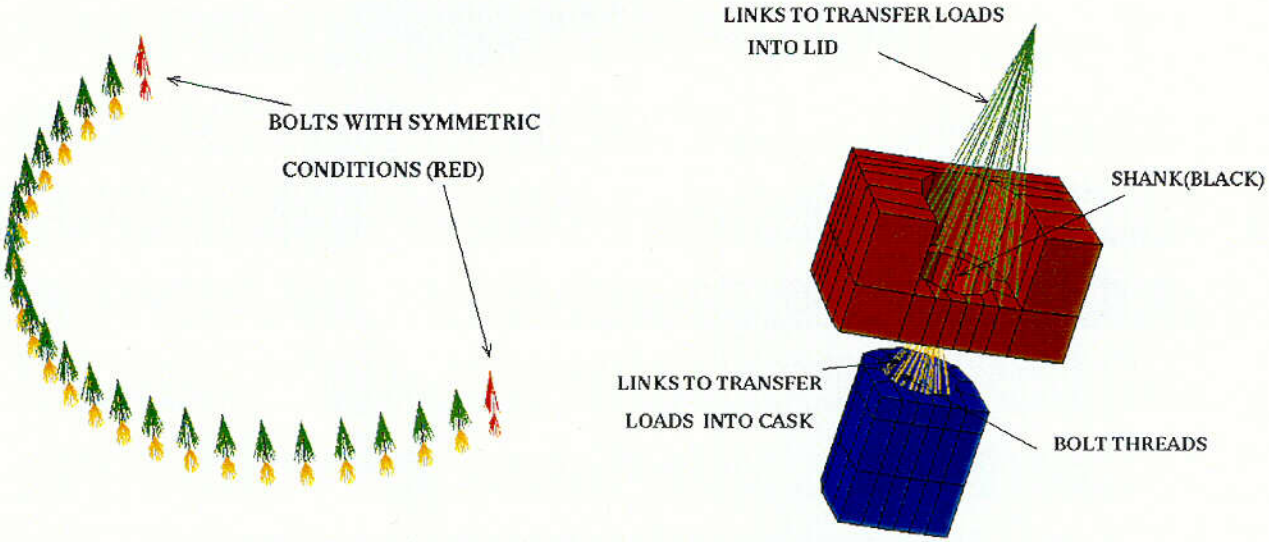


Figure 2.10.10-10: Details of Bolt Assembly LINK8 Elements

C04

The bottom cover and flange assembly is shown in Figure 2.10.10-11. This component is constructed out of SOLID45 elements and connects to the ram cover plate (Figure 2.10.10-12) through bolts represented by LINK8 elements. The ram cover plate is also connected at the side to the bottom plate by CONTACT52 elements. The bottom flange is connected to the inner and outer shell directly and to the lead shield through CONTACT173 and TARGET170 elements. Details of this assembly are shown below in Figure 2.10.10-13. There are 12 bolts (six in the half model) connecting the bottom plate to the ram cover plate. Details of the spider formation of the LINK8 elements are shown in Figure 2.10.10-13. A representation of the bolted joint joining the bottom plate to the ram cover plate is shown in Figure 2.10.10-14

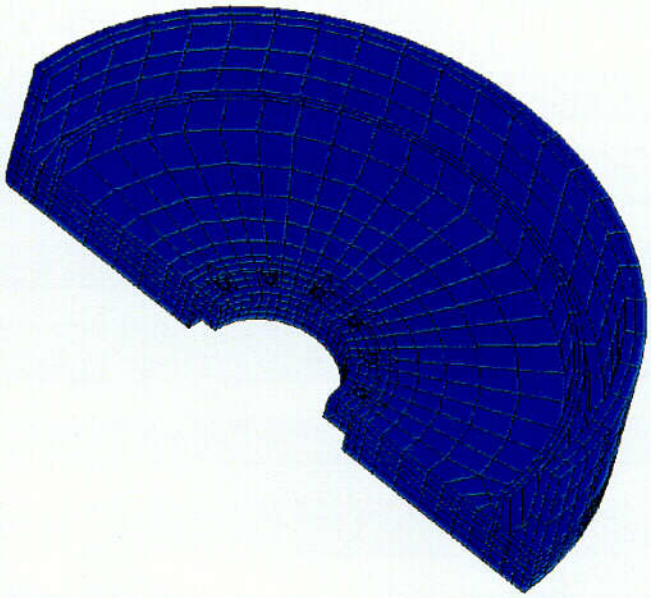


Figure 2.10.10-11: Cask Bottom Cover and Flange (3437 Elements and 4399 Nodes)

C05

Figure 2.10.10-12: Cask Bottom Ram Cover Plate

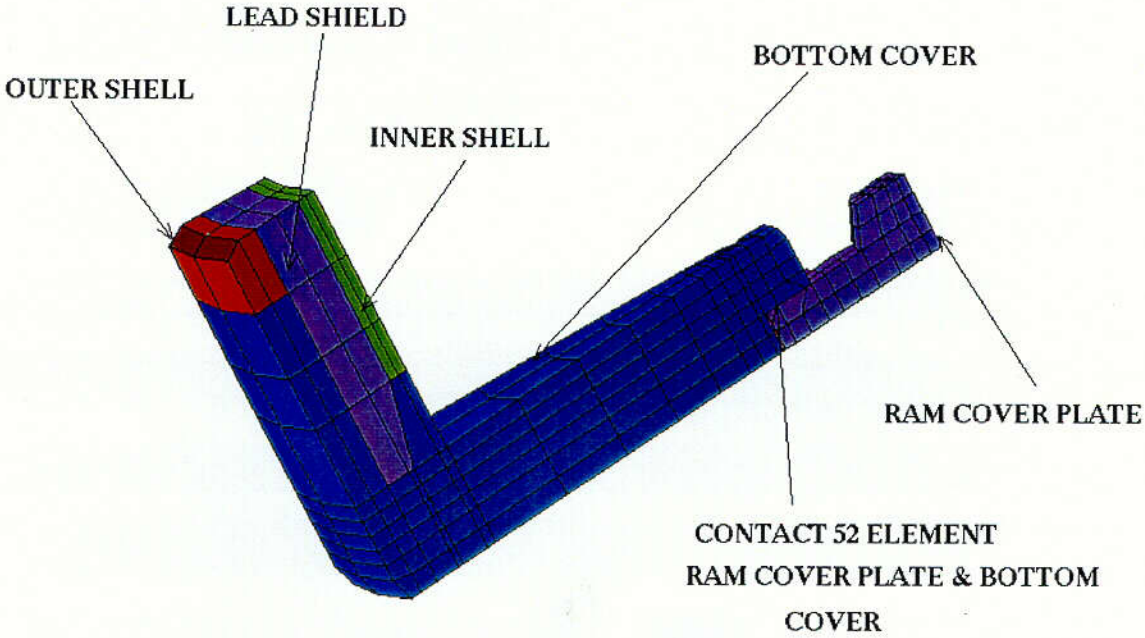


Figure 2.10.10-13. Segment of Cask at the Bottom Showing Assembly of Bottom Flange / Cover, Cam Cover Plate, Lead Shield, Inner and Outer Shell.
Also included is the location of a typical CONTACT 52 element.

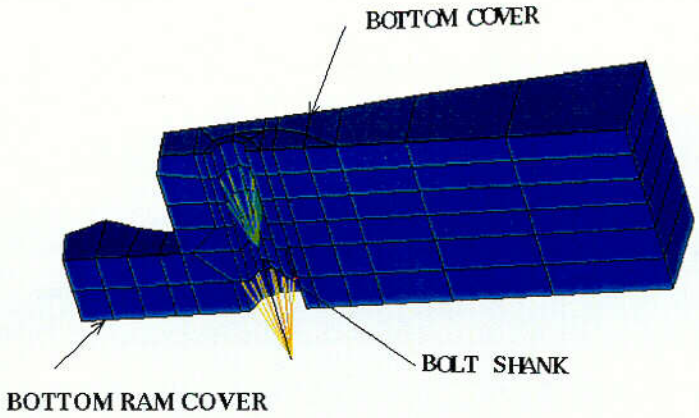


Figure 2.10.10-14. Cask Bottom and Ram Cover Bolted Joints

Fig 2.10.10-15 shows the details of the shear key bearing block and pad. This is attached to the outer shell with 48 coupled sets each in the x y and z directions around the edge. The aspect ratio of these elements is within acceptable limits.

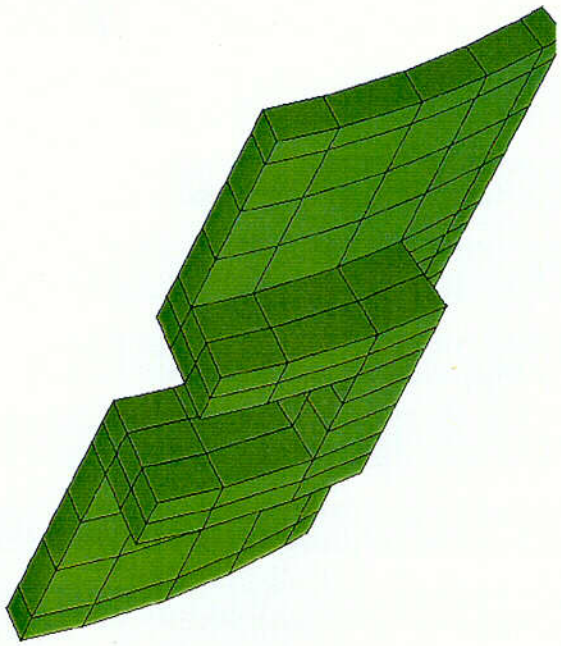


Figure 2.10.10-15: Shear Key Bearing Block and Pad (99 Elements and 240 Nodes)

2.10.10.3 Review of 3D FEM warning elements

Figure 2.10.10-16 shows a view of all warning elements output by the ANSYS program. The warning elements are divided into five groups: A, B, C, D and E. Detailed descriptions of the groupings are given below. The locations of these individual groups are shown with respect to the entire model. A total of 387 elements, out of 39,044, are indicated as warning elements (less than 1%).

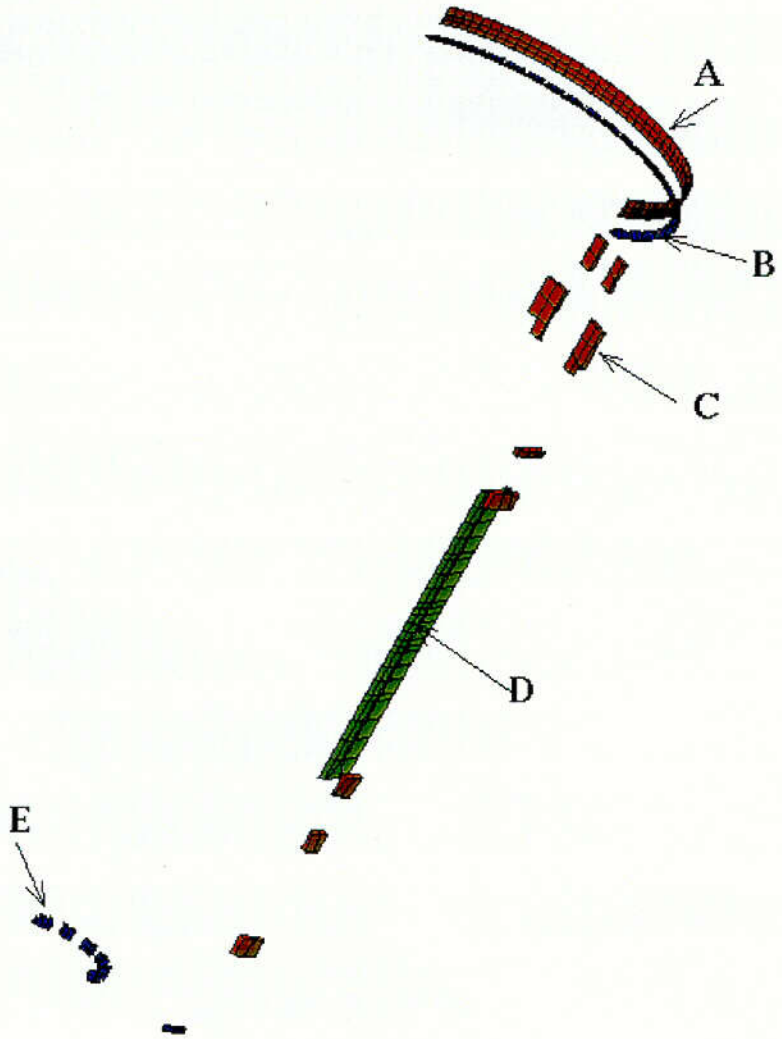


Figure 2.10.10-16: Warning Elements from the 3D FEM

C08

Figure 2.10.10-17 shows the elements of group A and their relative position in the lid. There are 144 elements in this set. The ANSYS warning messages (2 per element) is indicated as:

1. "Brick element has a pair of opposite edges that are 71.25 degrees away from being parallel. This exceeds the warning limit of 70 degrees."
2. "Brick element 14040 has an angle between adjacent edges of 167.3 degrees, which exceeds the warning limit of 155 degrees."

The interior angles differ from the set warning limits of 70 and 155 degrees. This distortion was due to modeling of the bolt holes. Since the warning elements are in a localized region and these warning factors are also close to the set limits, therefore they are deemed acceptable.

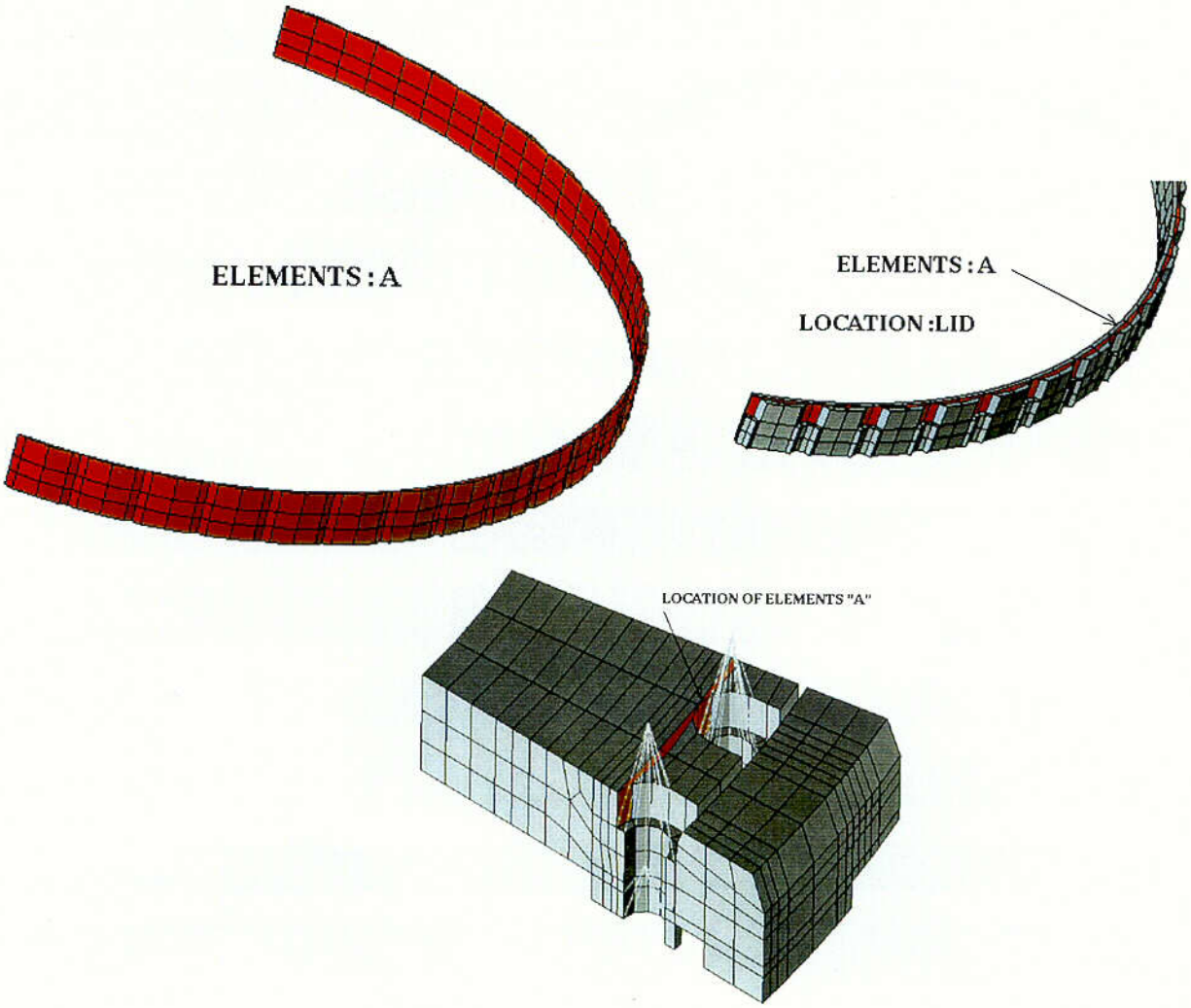


Figure 2.10.10-17: Warning Elements A from the 3D FEM in the Lid near the Bolt Holes

Figure 2.10.10-18 shows the details of the warning elements of group B. There are 96 elements and the ANSYS warning message is given as:

1. "Brick element has a pair of opposite edges that are 77.11 degrees away from being parallel. This exceeds the warning limit of 70 degrees."
2. "Wedge element has an aspect ratio of 24.38, which exceeds the warning limit of 20.

Since these warning factors are close to the set limits, therefore they are deemed acceptable.

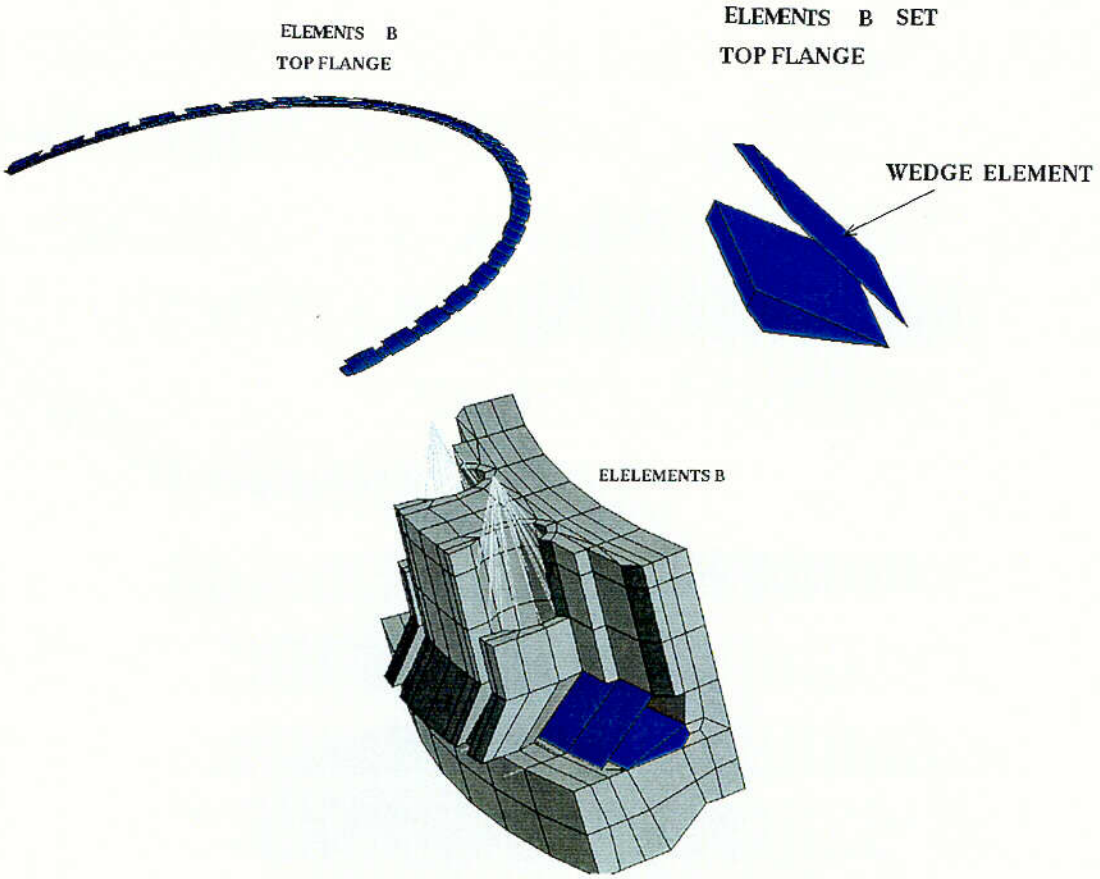


Figure 2.10.10-18: Warning Group B from the 3D FEM in Top Flange near the Bolt Holes

Details of group C elements are given in Figure 2.10.10-19. There are a total of 26 elements in this group. These elements are located in the outer shell (see figure). Distortions of these elements are due to the changes (circumferential shifts) made in the outer shell nodes to accommodate the shear key and the trunnions. The typical ANSYS warning messages given are as follows:

1. "Brick element has a face with a warping factor of 0.2036, which exceeds the warning limit of 0.2."
2. "Brick element has a pair of opposite edges that are 76.28 degrees away from being parallel. This exceeds the warning limit of 70 degrees."
3. "Brick element 8138 has an angle between adjacent edges of 159.9degrees, which exceeds the warning limit of 155 degrees."

These warning factors are very close to the limits set and deemed acceptable based on Engineering judgment.

Figure 2.10.10-19: Location of Warning Group C Elements in the Cask Outer Shell

There are 69 elements in group D of the warning element category. Sample ANSYS warning messages are indicated below:

1. "Brick element has an angle between adjacent edges of 170.7 degrees, which exceeds the warning limit of 155 degrees."
2. "Brick element has a pair of opposite edges that are 132.6 degrees away from being parallel. This exceeds the warning limit of 70 degrees."
3. "Brick element has an angle between adjacent edges of 162.3 degrees, which exceeds the warning limit of 155 degrees."
4. "Brick element has a pair of opposite edges that are 84.47 degrees away from being parallel. This exceeds the warning limit of 70 degrees.

The above warnings are generated in the shear key bearing block area. Since the stress generated in this region is calculated separately in SAR Section 2.5.2, therefore these warning are deemed acceptable.

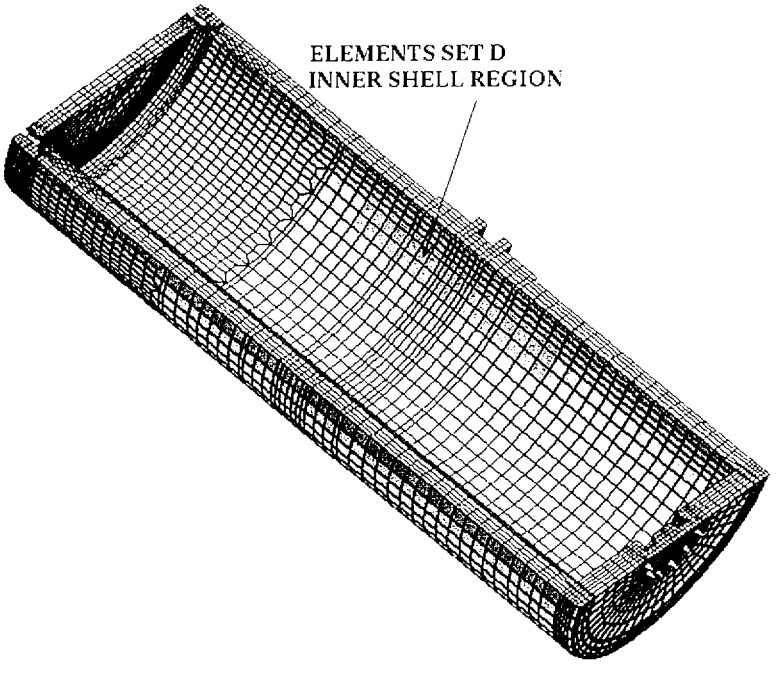


Figure 2.10.10-20: Warning Group D Elements in the Inner Shell Region

The set comprising warning group E elements is shown in Figure 2.10.10-21. There are 52 elements in this category. Three of these elements are in the bottom cover/flange region. Two of these had a warping factor of 0.4002, which was close to the error limit of 0.4. The details of the other elements in the ram cover plate are also shown. Distortions in this set of elements were due to modeling around the bolt holes. The ANSYS warning message was as follows:

1. "Brick element has a face with a warping factor of 0.2154, which exceeds the warning limit of 0.2."

These elements are deemed acceptable based on warning factors are very close to the default limits set, engineering judgment deems acceptable.

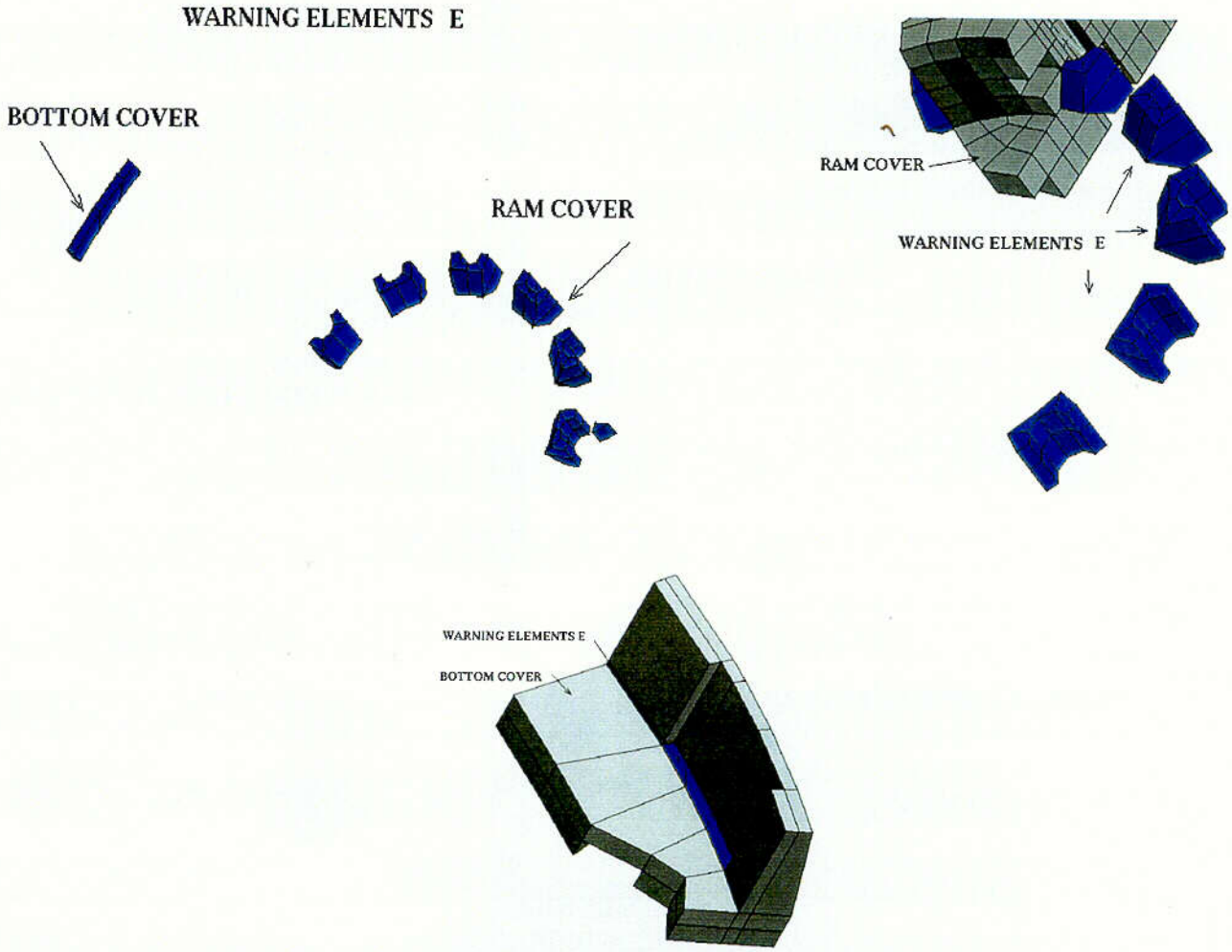


Figure 2.10.10-21: Warning Group E Elements in Bottom Plate / Flange and Ram Cover

2.10.10.4 Axisymmetric (2D) Finite Element Model Description

For all axisymmetric loading cases the 2D model used is shown in Figure 2.10.10-22. There are a total of 801 elements and 907 nodes in the model. The model is used to simulate the inner shell, outer shell, lead shield, lid, ram cover plate, bottom cover and top and bottom flanges as shown in Figure 2.10.10-22. The 3-D model of the NUHOMS®-MP197, described in Section 2.10.10.2, forms the basis for the 2D axisymmetric model construction.

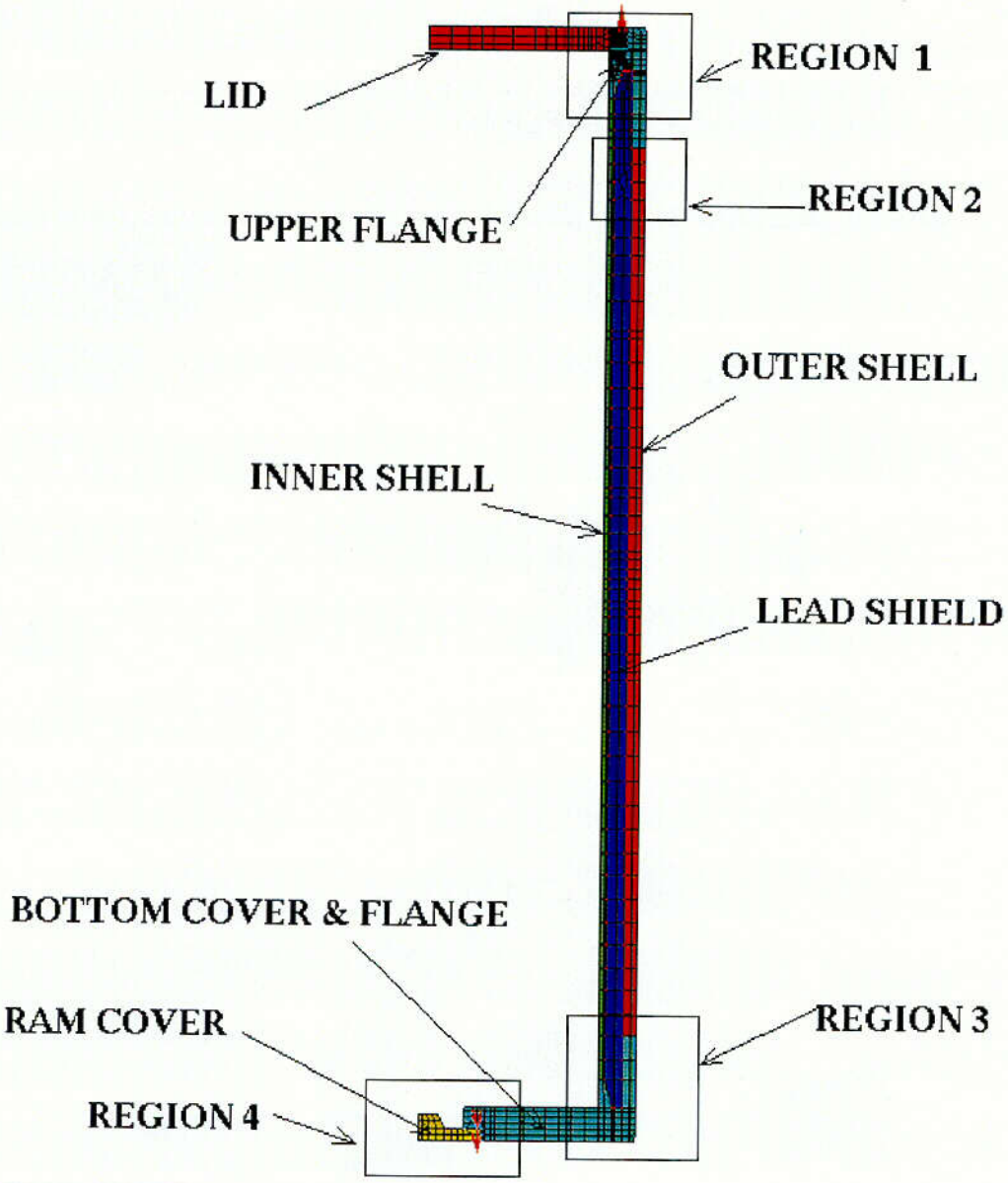


Figure 2.10.10-22: Axisymmetric Finite Element Model of the NUHOMS®-MP197 Transport Cask

The "surface effect" feature of ANSYS was used to create the PLANE42 axisymmetric elements through the thickness preserving the mesh density of the 3D model. The lid, upper flange, inner shell, outer shell, bottom flange, bottom cover and ram cover were modeled with PLANE42 elements. The surface elements TARGET170 and CONTACT173 in the 3D model were replaced with CONTACT12 elements. These CONTACT12 elements connect on a point-to-point basis. The four regions shown in Figure 2.10.10-22 are described below.

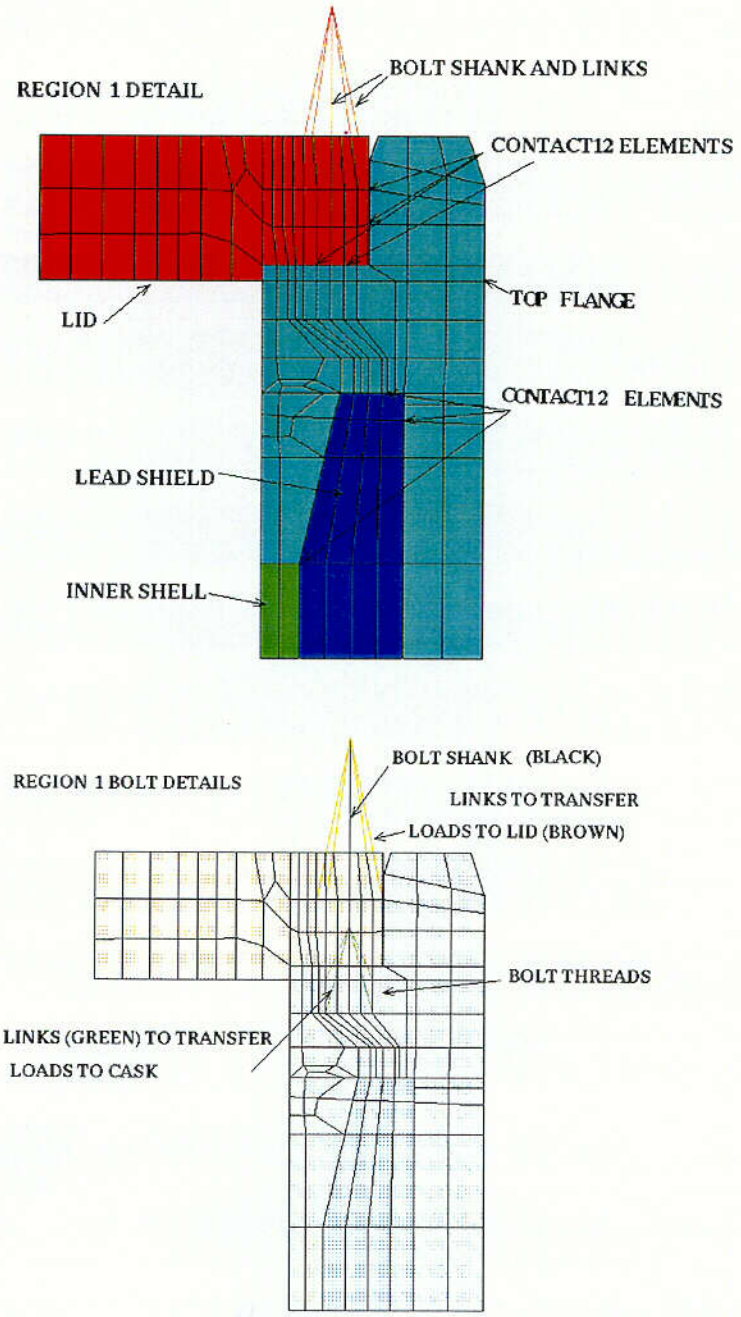


Figure 2.10.10-23a & b: Region 1 Showing Modeling Detail and Bolt Idealization

Figure 2.10.10-23a and b show the details of the cask in the area indicated as region 1. The color scheme distinguishes the various components of the cask in this region. The locations near the lead shield where the CONTACT173 and TARGET 170 elements were replaced by CONTACT12 elements are shown. The other set of CONTACT12 elements connecting the lid to the top flange is also indicated. Figure 2.10.10-23b shows a transparent mesh giving details of a single bolt connection. Two sets of LINK1 elements (brown and green) connect the bolt shank to the lid and top flange.

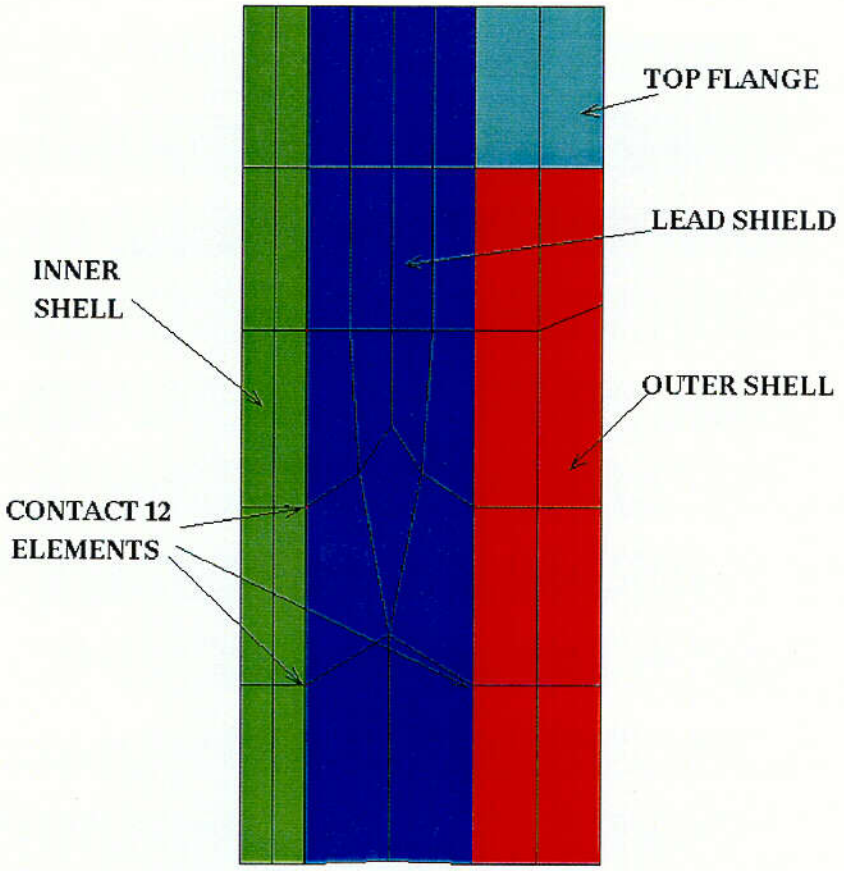


Figure 2.10.10-24: Details of Region 2

Figure 2.10.10-24 shows the details of the assembly in region 2. The transition 2 to 4 elements in the lead shield can be seen. This is done to accommodate the detailed mesh near the top flange. The lead shield is isolated from the inner shell, outer shell and the flanges using CONTACT12 elements. Some locations of the CONTACT12 elements are also shown. This region is very similar to the 3D mesh.

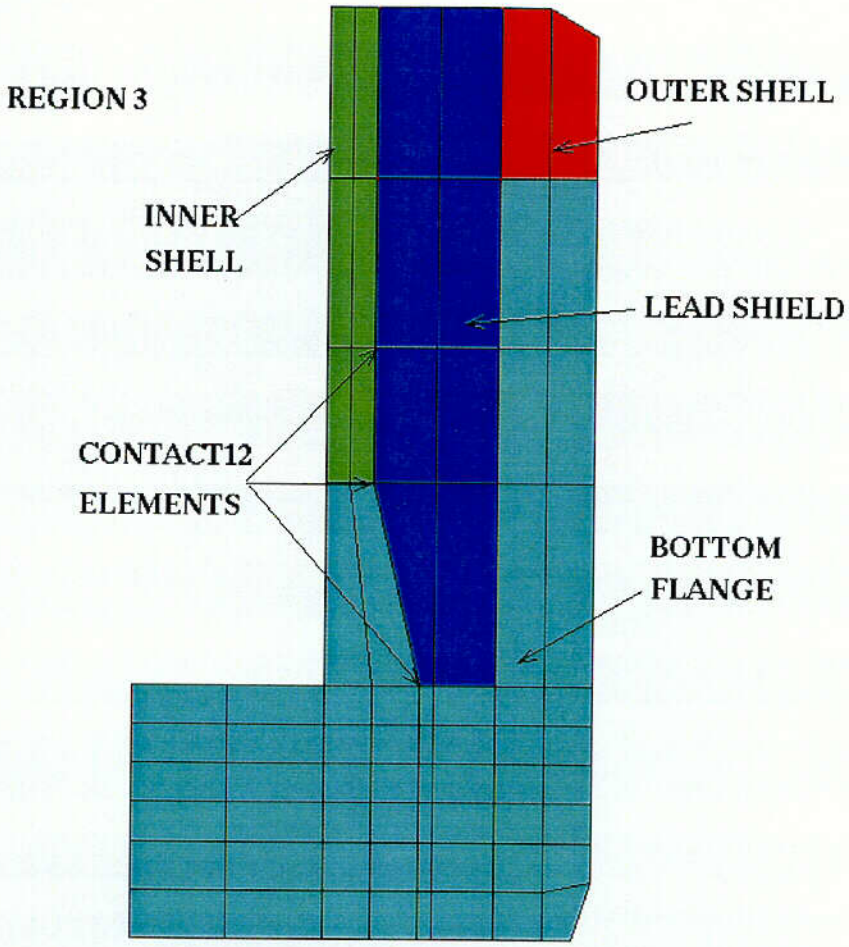
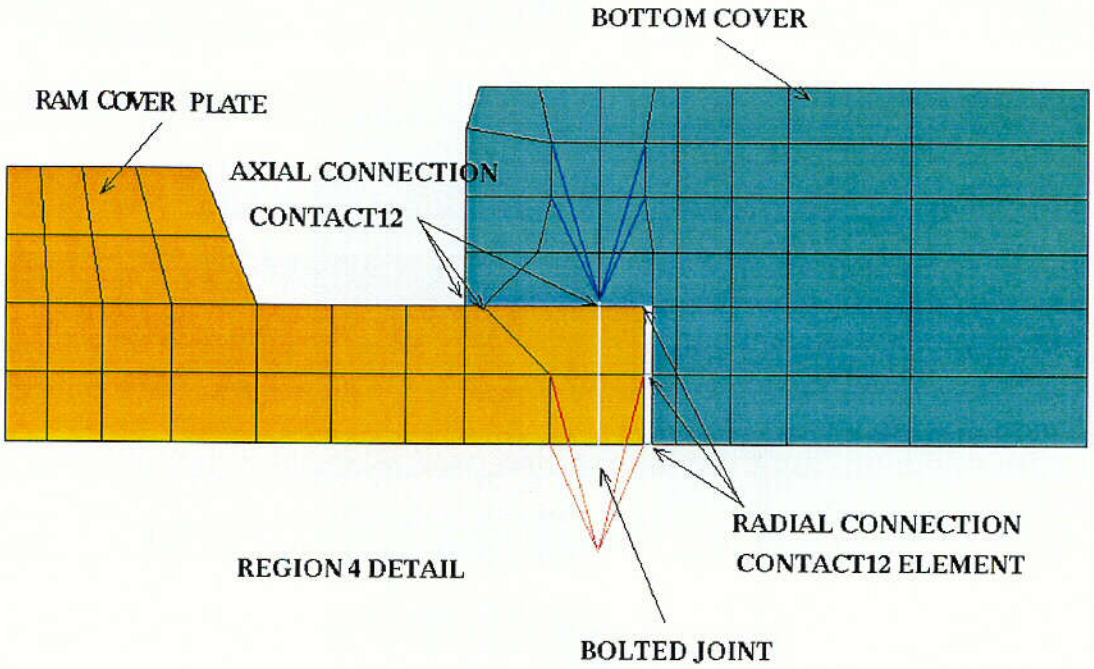


Figure 2.10.10-25: Region 3 Showing Finite Element Idealization near the Bottom Flange

A detailed view of the finite element idealization near the bottom flange is shown in Figure 2.10.10-25. Also shown are some of the CONTACT12 elements that separate the lead shield and the rest of the structure. The taper in the lead shield near the bottom has been retained from its 3D counterpart. All four noded elements are axisymmetric PLANE42 from the ANSYS finite element library.

CONTACT12 elements and bolted joint connecting the ram cover to the bottom flange are shown in figures 2.10.10-26a and b. This is the area marked Region 4 in Figure 2.10.10-22. The axial and radial contact locations (CONTACT12) are indicated in Figure 2.10.10-26a. The transparent mesh of figure 2.10.10-26b shows the details of the LINK1 assembly used in the bolted joint idealization, which is similar to the lid/ top flange connection.



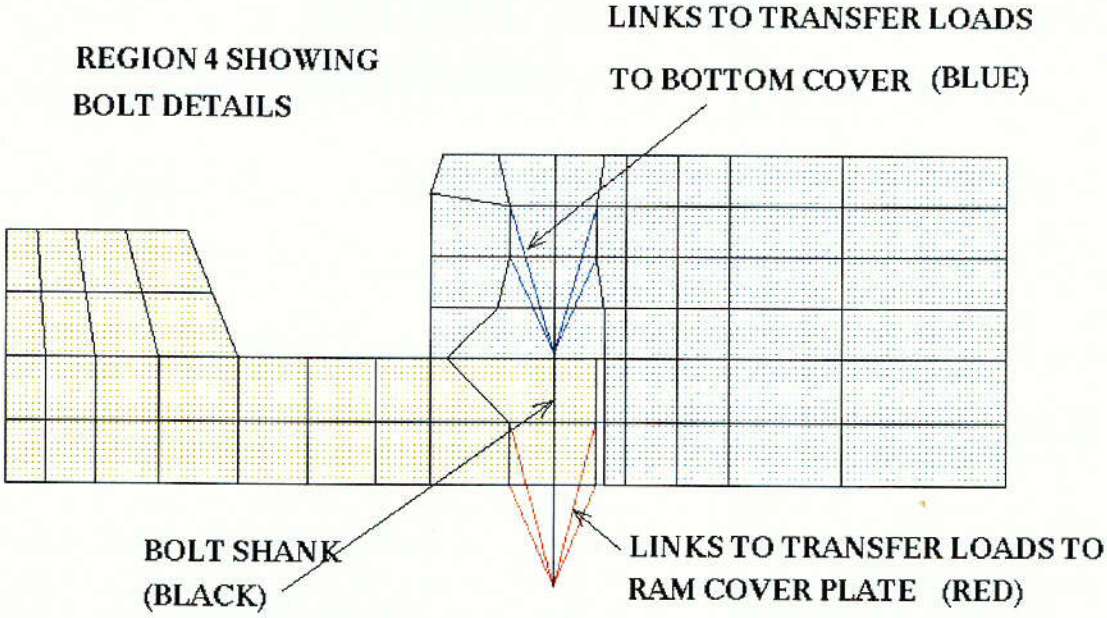


Figure 2.10.10-26a & b: Region 4 Showing Ram Cover, Bottom Flange Detail

2.10.10.5 Review of 2D FEM Warning Elements

An elements check command yielded two sets of warnings.

1. All CONTACT12 elements not coincident gave the warning:

    i. "Nodes I and J of element (CONTAC12) are not coincident".

2. The three PLANE42 warning elements (Region 1) are shown in Figure 2.10.10-27. The warning messages as follows:

    i. "Quadrilateral element has a pair of opposite edges that are 95.83 degrees away from being parallel. This exceeds the warning limit of 70 degrees."
    ii. "Quadrilateral element has an angle between adjacent edges of 162 degrees, which exceeds the warning limit of 155 degrees."
    iii. "Quadrilateral element has a pair of opposite edges that are 95.7 degrees away from being parallel. This exceeds the warning limit of 70 degrees."
    iv. "Quadrilateral element has an angle between adjacent edges of 161.7 degrees, which exceeds the warning limit of 155 degrees. "

Since the 3 warning elements are in a localized region and stress results are not critical at this region, therefore the elements warning are deemed acceptable based on Engineering judgment.

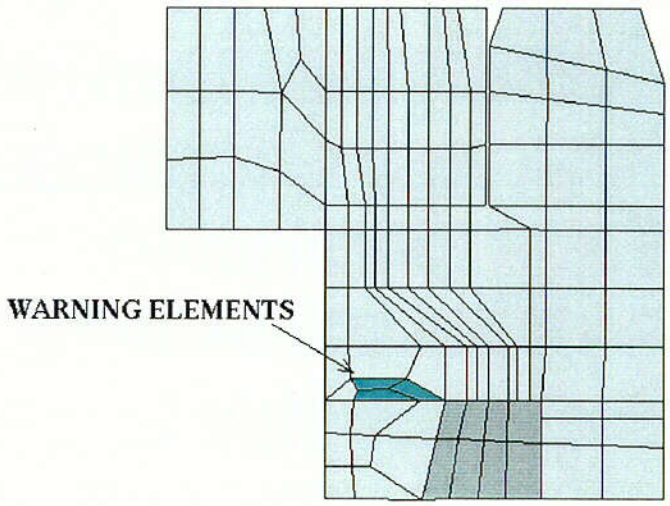


Figure 2.10.10-27: Warning Elements (Region 1) in Top Flange Area

2.10.10.6 Maximum Component Stress Evaluation

2.10.10.6.1 Approach

This section evaluates the maximum stresses that occur in each major component of the NUHOMS®-MP197 Transport Cask during all normal and hypothetical accident conditions of transport.

This analysis employs the same finite element model, loading conditions and results described in SAR Appendix 2.10.1. No new finite element analysis runs are made for this section. This analysis provides a more detailed evaluation of the stress analysis results generated in SAR Appendix 2.10.1. In particular, the maximum stress and the overall stress distribution in each major component of the transport cask are extracted from the ANSYS results for each normal and hypothetical accident condition load combination. The overall stress distributions are plotted, and the maximum component stresses and are tabulated and compared to allowable values.

The NUHOMS®-MP197 Transport Cask is separated into the following three major components, for stress reporting purposes.

HUOMS®-MP197 Transport Cask Components

Component	Material
Lid	SA-693 Type 630 H1100
Body Containment (Bottom Plate, Ram Port Cover, Flange, and Inner Shell)	SA-240 Type XM-19
Outer Shell	SA-240 Type 316

An element plot of each major component listed above is provided in Figures 2.10.10-28, 2.10.10-29, and 2.10.10-30 of this Appendix.

For each load combination the maximum stress in each of these components is extracted from the result files and compared to the allowable values for each material.

The load combinations evaluated (runs 13 through 42) are described in SAR Appendix 2.10.1 and listed in Tables 2.10.10-2 and 2.10.10-3 of this Appendix. These load combinations are generated through the combination of the twelve individual loads also described in SAR Appendix 2.10.1 and listed in Table 2.10.10-1 of this Appendix.

2.10.10.6.2 Stress Reporting Methodology for 3-Dimensional Model

Within the 3-dimensional finite element model there are a few locations where boundary conditions or contact elements generate spurious local high stresses. When reporting the component maximum stresses, the elements at the following locations are unselected so that the unrealistic stresses are not reported. Stresses at these unselected locations are evaluated separately.

Point Displacement Boundary conditions

For several loading conditions, a displacement boundary condition was placed on a single node of the large 3D model. This boundary condition is used to react any unbalanced global forces in the model. Because of the complicated three-dimensional nature of the finite element model, it is difficult to balance global forces and accelerations perfectly. Consequently, small residual unbalanced forces act as a point load at the support location. This type of loading generates extremely high and localized stresses that are fictitious. In order to eliminate this incorrect stress, elements attached to the supported node are unselected.

Contact Elements

Node-to-node contact elements are used at the intersection between the top surface of the transport cask flange and the bottom surface of the lid. Due to the their nonlinear and node-to-node nature, these elements can sometimes generate unrealistic bearing stresses. This is because they do not evenly distribute the reaction forces generated between the flange and the lid evenly. Also, since link elements are used to model the closure lid bolts, the modeled bolts are only capable of generating tensile forces and not bending moments. Consequently the moments in the lid generated during load cases such as the end drop or corner drop on the lid, must be reacted by concentrated forces couple in the lid bolts and flange. Therefore, additional unrealistic bearing forces are generated in the flange. For these reasons, elements attached to the contact elements are unselected, so that no spurious stresses are reported. The bearing stresses on the top surface of the flange and shear stress in flange thread are evaluated by hand calculation separately in Section 2.10.10.7 of this Appendix.

Bearing Block

Under certain loading conditions, such as rail car shock and vibration, the transport cask model is supported in the longitudinal direction at the shear key. Axial displacement boundary conditions are placed on a few nodes on the inner surface of the bearing block. This boundary condition is relatively localized, and results in unreasonably high local stresses. Consequently, the bearing block elements are unselected. A separate hand calculation, in SAR Section 2.5.2, evaluates the stresses in the bearing block and in the outer shell.

2.10.10.6.3 Results

For each load combination listed in Tables 2.10.10-2 and 2.10.10-3 of this Appendix, the maximum stress intensity for each component is extracted from the result files. These stress intensities are listed in Tables 2.10.10-4 and 2.10.10-5 of this Appendix for normal and hypothetical accident condition load combinations respectively. A stress intensity plot is also provided for each load case in order to give an overall description of the stress distribution throughout the transport cask (see Figures 2.10.10-31 through 2.10.10-60 of this Appendix)

The maximum component nodal stress intensities are compared to the allowable membrane stress intensity given in SAR Appendix 2.10.1. If the maximum nodal stress intensity exceeds the membrane allowable, the nodal stresses are linearized through the thickness of the component. Both linearized membrane and membrane plus bending stress intensities are then reported and compared to allowable membrane and membrane plus bending stress intensities respectively (see Table 2.10.10-4 and 2.10.10-5 of this Appendix). For all load combinations, the maximum stresses in the NUHOMS-61B transport Cask are less than their corresponding allowable stresses.

2.10.10.7 Evaluation of Bearing Stresses in Flange

2.10.10.7.1 Approach

The stress analysis of the NUHOMS®-MP197 Transport Cask for normal and accident conditions is described in SAR Appendix 2.10.1. The cask lid bolt and bolt head are represented with ANSYS LINK1 elements in 2D model and LINK8 in 3D model. The Link elements are uniaxial tension-compression elements with no bending capabilities. Also the thread connections between the bolt and flange were modeled by link elements. Since these links were modeled at a few nodes only, this resulted in high local stresses. The purpose of this evaluation is to calculate the flange thread shear stresses and lid / flange bearing stress based on the actual bolt engagement length and bearing area. The stress evaluation is conducted both for normal and accident conditions.

Link elements are adequate to transfer axial tension or compression loads. However, since they are connected between a few nodes of the model to simulate flange-thread connection, it resulted in high local flange stresses at the connection nodes. In reality, the bolt thread connection is continuous at the thread contact. Further, since link elements were not capable of taking any bending load, entire bending had to be resisted by the perforated lid and the flange. In order to obtain realistic results, the flange thread shear stress and lid / flange bearing stress are calculated here by using the actual thread geometry and the bearing areas. The bolt forces and the lid-flange contact forces, for the stress calculations are, however, taken from the ANSYS result files in Reference 3.

This calculation is based on the results of the ANSYS runs described in SAR Appendix 2.10.1. Therefore, assumptions used in that Appendix are also applicable to this evaluation. All material properties are taken from ASME B&PV Code Section II, Part D [2] at 300°F. The bolt hole areas are subtracted to obtain the net bearing area for the uniformly distributed loads.

2.10.10.7.2 Loading Conditions

All load combinations for the Normal and the Accident Conditions are listed in Tables 2.10.10-2 and 2.10.10-3 of this Appendix. A screening of all load combinations indicated that for the maximum flange thread shear stress and lid-flange bearing stress, it will suffice to analyze the following load combinations. The stress results from these selected load combinations will envelop the stress results from all load combinations.

Normal Conditions

A. Preload + Internal pressure (50 psi) + Thermal hot + Rail car Shock (Run 19)
B. Preload + Ext. pressure (25 psi) + Thermal cold + 1 ft. Drop on Lid end (Run 22)

Accident Conditions

C. Preload + Ext. pressure (25 psi) + Thermal cold + 30 ft. Drop on Lid end (Run 30)
D. Preload + Ext. pressure (25 psi) + Thermal cold + 30 ft. Drop on C.G.-Over-Corner (Run 36)

2.10.10.7.3 Normal Condition Stresses

In order to calculate the shear stresses the flange threads and flange bearing stresses, the maximum bolt force and contact forces for each load case are obtained using the ANSYS postprocessor with the appropriate result file.

A. Preload + Internal Pressure (50 psi) + Thermal Hot + Rail Car Shock

Maximum Shear Stress in Flange Threads

From the ANSYS results file, the maximum lid bolt force = 128,200 lb.

The cask flange (SA-240, Gr. XM-19, S_u = 100.0 ksi. at room temperature) threads have helicoil inserts (1185-24CN-2250) of Type 304, stainless steel wire (ultimate strength at room temperature is 200 to 250 ksi, [3]). The lid bolts are constructed from SA-540 Gr. B24 (S_u = 165 at room temperature). The flange material is, therefore, weaker than the helicoil and bolt materials.

A_n = shear area of internal threads = $3.1416\, nL\, D_{s\,min}\, [1/(2n) + .57735(D_{s\,min} - E_{n\,max})]$[4]

For the tapped hole for Helicoil insert connection:

$E_{n\,max}$ = maximum pitch diameter of internal threads = 1.6177 in. [3]
$D_{s\,min}$ = minimum major diameter of external threads = 1.702 in. [3]
L = engagement length = 2.25 in.
n = threads/inch = 6

Therefore,

$A_n = 3.1416(6)(2.25)(1.702)[1/(2\times6) + .57735\,(1.702 - 1.6177)] = 9.528$ in.^{2}
Max. Shear Stress = 128,200/9.528 = 13,455 psi.

For SA-240, Gr. XM-19, S_m = 31.4 ksi. at 300°F. Therefore, the allowable shear stress = $0.6S_m$ = 0.6 (31.4) = 18.84 ksi. > 13.46 ksi … o.k.

Bearing Stress between Flange and Lid

From the ANSYS results file for run 19, the total contact force on the 180° model = 2,797,309 lbs. This total force is computed by summing all normal forces in the contact elements between the flange and the lid. Therefore, the contact force for full lid and flange = 2 × 2,797,309 = 5,594,618 lbs.

From SAR Appendix 2.10.2, the lid-bolt preload (maximum torque) = 120,800 lbs. Therefore, the total bolt force = 48 × 120,800 = 5,798,400 lbs. Thus, there is only a small loss of preload contact force due to pressure, thermal and rail car shock loads, and the contact forces are quite uniformly distributed over the entire flange-lid bearing area.

The bearing area is computed between the lid shoulder diameter (72.31 in.) and the flange diameter (68.42 in.). 48-lid bolt half-hole areas are subtracted to get the net bearing area.

$$\text{Net Bearing Area} = \pi/4\,(72.31^2 - 68.42^2) - \pi/4 \times \tfrac{1}{2}\,(1.69^2) \times 48$$
$$= 429.959 - 53.836 = 376.123 \text{ in.}^2$$

Bearing Stress = 5,594,618 / 376.123 = 14,874 psi

The flange is constructed from SA-240, Gr. XM-19, which has a larger yield strength than that of the lid (SA-705, Type 630).

Allowable Bearing Stress = S_y, Flange = 43.3 ksi. > 14.87 ksi … o.k.

B. <u>Preload + External Pressure (25 psi) + Thermal Cold + 1 Foot Lid Drop</u>

Maximum Shear Stress in Flange Threads

From the ANSYS results file (Run 22), the lid bolt force in 48 bolts = 5,920,600 lb. Therefore, the force in one bolt = 5,920,600/48 = 123,346 lb.

Thread Shear Area = 9.528 in.^{2}
Shear Stress = 123,346/9.528 = 12,946 psi.

Allowable Shear Stress = $0.6S_m$ = 0.6 (31.4) = 18.84 ksi. > 12.95 ksi … o.k.

Bearing Stress between Flange and Lid

From the ANSYS results file for run 22, the total contact force = 5,994,840 lbs.

Net Bearing Area = 376.123 in.^{2}
Bearing Stress = 5,994,840 / 376.123 = 15,939 psi

Allowable Bearing Stress = S_y, Flange = 43.3 ksi. > 15.94 ksi … o.k.

2.10.10.7.4 Accident Condition Stresses

C. Preload + External Pressure (25 psi) + Thermal Cold + 30 Foot Lid Drop

Maximum Shear Stress in Flange Threads

From the ANSYS results file, the lid bolt force in 48 bolts = 5,950,400 lb. Therefore, the force in one bolt = 5,950,400/48 = 123,967 lb.

Thread Shear Area = 9.528 in.^{2}
Shear Stress = 123,967/9.528 = 13,011 psi

Allowable Shear Stress = 0.42 S_u = 0.42 (94.2) = 39.56 ksi. > 13.01 ksi ... o.k.

Bearing Stress between Flange and Lid

From the ANSYS results file for run 30, the total contact force = 6,006,630 lbs.

Net Bearing Area = 376.123 in
Bearing Stress = 6,006,630 / 376.123 = 15,970 psi.

Allowable Bearing Stress = S_u, Flange = 94.2 ksi. > 15.97 ksi... o.k.

D. Preload + External Pressure (25 psi) + Thermal Cold + 30 ft. CG Over Corner Drop

Maximum Shear Stress in Flange Threads

From the ANSYS results file (Run 19), the maximum lid bolt force = 174,790 lb.

Thread Shear Area = 9.528 in^2
Max. Shear Stress = 174,790/9.528 = 18,345 psi.

Allowable Shear Stress = 0.42S_u = 0.42 (94.2) = 39.56 ksi. > 18.35 ksi... o.k.

Bearing Stress between Flange and Lid

From the ANSYS results file for run 19, the total contact force on the 180° model = 8,563,485 lbs. Therefore, the contact force for full lid and flange = $2 \times 8{,}563{,}485 = 17.127 \times 10^6$ lb.

Due to the nature of the CG over corner drop event, this contact force is comprised of the following two components.

- A uniform distributed force due to the bolt preload and the 25 psi external pressure load.
- A non-uniform contact force due to the corner dorp impact acceleration forces.

The uniform distributed force acts on the entire net bearing area while the non-uniform force is assumed to be distributed only on half of the net bearing area.

Uniform Distributed Load Bearing Stress:

Total Bolt Preload Force = $120{,}840 \times 48 = 5.800 \times 10^6$ lb. (Appendix 2.10.2)
25 psi. Pressure Force (on 74.68" diameter Lid) = $\pi/4\ (74.68^2) \times 25 = 0.109 \times 10^6$ lb.
Total Uniform Bearing Load = $5.800 \times 10^6 + 0.109 \times 10^6 = 5.909 \times 10^6$ lb.
Bearing Stress = $5.909 \times 10^6\ /\ 376.123\ = 15{,}710$ psi.

Non- Uniform Load Bearing Stress:

The non-uniform contact load is the difference between the total contact force from the ANSYS results file and the uniformly distributed force computed above.

Non-Uniform Bearing Force = $17.127 \times 10^6 - 5.909 \times 10^6\ = 11.218 \times 10^6$ lb.
Bearing Area = 50% (376.123) = 188.06 in^2
Bearing Stress = $11.218 \times 10^6\ /\ 188.06 = 59{,}651$ psi.

Total Bearing Stress = 15,710 + 59,651 = 75,361 psi.

Allowable Bearing Stress = S_u, Flange = 94.2 ksi > 75.36 ksi… o.k.

2.10.10.7.5 <u>Conclusion</u>

From the above analysis, it is concluded that the maximum shear stresses in the flange threads and the maximum bearing stresses between lid and flange are within the code allowables for normal and accident load conditions. Consequently, the NUHOM®-MP197 lid and flange connection is structurally adequate.

2.10.10.8 References

1. ANSYS Computer Code and User's Manuals, Volumes 1-4, Rev. 5.6.
2. American Society of Mechanical Engineers, ASME Boiler and Pressure Vessel Code, Section II, Part D, 1998, including 1999 Addenda.
3. Helicoil Catalog, Bulletin 1000, Rev.4/98,"Screw Thread Inserts, Inch Series, Design & Production Manual".
4. Machinery Handbook, 21st Edition, Industrial Press, 1979.

Table 2.10.10-1
Individual Load Conditions

Run No.	Applicable Individual Loads	Load Used in Run
1	Bolt preload	-
2	Internal pressure	50 psig
3	External pressure	25 psig
4	Thermal stresses in 100°F hot ambient environment	-
5	Thermal stresses at –20° F cold ambient environment	-
6	3G lifting	3G
7	Rail Car Shock loads	4.7G – all directions
8	Rail car vibration loads	0.37G – vertical 0.19G –lateral 0.19G – longitudinal
9	1 foot end drop on lid end	30 G
10	1 foot end drop on bottom end	30 G
11	1 foot side drop	30 G
12	1G gravity loading	1 G

Table 2.10.10-2
Normal Condition Load Combinations

Run No.	Load Combination	Applicable Individual Loads Applied in the ANSYS Model								
		Bolt Pre-load	Gravity 1g	Int. Pres.	Ext. Pres.	Thermal Hot	Thermal Cold	Thermal -40°F Uniform	Rail Car Vib.	Rail Car shock
13	Hot Environment (100° F amb.)	x	x	x		x				
14	Cold Environment (-40° F amb.)	x	x		x			x		
15	Increased External Pressure	x	x		x		x			
16	Min. External Pressure	x	x	x		x				
17	Rail Car Vibration	x		x		x			x	
18		x			x		x		x	
19	Rail Car Shock	x		x		x				x
20		x			x		x			x

Run No.	Load Combination	Applicable Individual Loads Applied in the ANSYS Model							
		Bolt Pre-load	Internal Pres. (50 psi)	External Pres. (25 psi)	Thermal Hot	Thermal Cold	Lid End Drop	Bottom End Drop	Side drop
21	1 Ft End Drop on Lid End	x	x		x		x		
22		x		x		x	x		
23	1 Ft End Drop on Bottom End	x	x		x			x	
24		x		x		x		x	
25	1 Ft Side Drop	x	x		x				x
26		x		x		x			x

Table 2.10.10-3
Hypothetical Accident Condition Load Combinations

Run No.	Load Combination	Applicable Individual Loads Applied in the ANSYS Model							
		Bolt Pre-Load	Int. Pres. (50 psi)	Ext. Pres. (25 psi)	Thermal Hot	Thermal Cold	Lid End Drop	Bot. End Drop	Side Drop
27	30 Ft. End Drop on Bottom End	x	x		x			x	
28		x		x		x		x	
29	30 Ft. End Drop on Lid End	x	x		x		x		
30		x		x		x	x		
31	30 Ft. Side Drop	x	x		x				x
32		x		x		x			x

Run No.	Load Combination	Applicable Individual Loads Applied in the ANSYS Model							
		Bolt Pre-Load	Int. Pres. (50 psi)	Ext. Pres. (25 psi)	Thermal Hot	Thermal Cold	Corner Drop Lid End	Corner Drop Bot End	Oblique Drop Lid End
33	30 Ft. CG Over Corner Drop on Bottom End	x	x		x			x	
34		x		x		x		x	
35	30 Ft. CG Over Corner Drop on Lid End	x	x		x		x		
36		x		x		x	x		
37	30 Ft. 20° Oblique Impact on Lid End	x	x		x				x
38		x		x		x			x

Table 2.10.10-3
Hypothetical Accident Condition Load Combinations
(Continued)

Run No.	Load Combination	Applicable Individual Loads Applied in the ANSYS Model						
		Bolt Pre-load	Int. Pres. (50 psi)	Ext. Pres. (25 psi)	Thermal Hot	Thermal Cold	Oblique Drop Lid End	Oblique Drop Bottom End
39	30 Ft. 20° Oblique Impact on Bottom End	x	x		x			x
40		x		x		x		x

Run No.	Load Combination	Applicable Individual Loads Applied in the ANSYS Model						
		Bolt Pre-load	Int. Pres. (50 psi)	Ext. Pres. (25 psi)	Thermal Hot	Thermal Cold	Immersion Ext. Pres. (290 psi)	Fire
41	Immersion (290 psi)	x				x	x	
42	Fire Accident	x	x					x

Table 2.10.10-4
Maximum Component Stresses for Normal Conditions

Run No.	Description	Location	Maximum Stress Intensity (ksi)	Stress Type	Allowable Stress (ksi)**	Linearized nodes	Figure
13	Hot Environment (100° F amb.)	Lid	30.5	$P_m + P_b$	46.7		2.10.10-31
		Body Containment	17.7	$P_m + P_b$	31.4		
		Outer Shell	14.8	$P_m + P_b$	20.0		
14	Cold Environment (-40° F amb.)	Lid	27.9	$P_m + P_b$	46.7		2.10.10-32
		Body Containment	13.2	$P_m + P_b$	31.4		
		Outer Shell	12.3	$P_m + P_b$	20.0		
15	Increased External Pressure	Lid	27.5	$P_m + P_b$	46.7		2.10.10-33
		Body Containment	13.7	$P_m + P_b$	31.4		
		Outer Shell	12.5	$P_m + P_b$	20.0		
16	Min. External Pressure	Lid	30.5	$P_m + P_b$	46.7		2.10.10-34
		Body Containment	17.7	$P_m + P_b$	31.4		
		Outer Shell	14.8	$P_m + P_b$	20.0		
17	Rail Car Vibration, Hot Environment	Lid	30.3	$P_m + P_b$	46.7		2.10.10-35
		Body Containment	17.4	$P_m + P_b$	31.4		
		Outer Shell	15.2	$P_m + P_b$	20.0		
18	Rail Car Vibration, Cold Environment	Lid	27.2	$P_m + P_b$	46.7		2.10.10-36
		Body Containment	14.0	$P_m + P_b$	31.4		
		Outer Shell	12.9	$P_m + P_b$	20.0		
19	Rail Car Shock, Hot Environment	Lid	36.3	$P_m + P_b$	46.7		2.10.10-37
		Body Containment	26.0	$P_m + P_b$	31.4		
		Outer Shell	10.4	P_m	20.0	67 – 78	
			20.3	$P_m + P_b$	30.0		
20	Rail Car Shock, Cold Environment	Lid	33.6	$P_m + P_b$	46.7		2.10.10-38
		Body Containment	24.3	$P_m + P_b$	31.4		
		Outer Shell	12.0	P_m	20.0	76 – 78	
			20.3	$P_m + P_b$	30.0		
21	1 Ft Lid End Drop, Hot Environment	Lid	12.4	$P_m + P_b$	46.7		2.10.10-39
		Body Containment	20.2	$P_m + P_b$	31.4		
		Outer Shell	13.7	$P_m + P_b$	20.0		
22	1 Ft Lid End Drop, Cold Environment	Lid	12.6	$P_m + P_b$	46.7		2.10.10-40
		Body Containment	18.6	$P_m + P_b$	31.4		
		Outer Shell	11.6	$P_m + P_b$	20.0		
23	1 Ft Bottom End Drop, Hot Environment	Lid	12.3	$P_m + P_b$	46.7		2.10.10-41
		Body Containment	17.1	$P_m + P_b$	31.4		
		Outer Shell	13.8	$P_m + P_b$	20.0		
24	1 Ft Bottom End Drop, Cold Environment	Lid	14.0	$P_m + P_b$	46.7		2.10.10-42
		Body Containment	18.8	$P_m + P_b$	31.4		
		Outer Shell	11.7	$P_m + P_b$	20.0		
25	1 Ft Side Drop, Hot Environment	Lid	38.3	$P_m + P_b$	46.7		2.10.10-43
		Body Containment	20.3	P_m	31.4	281 – 276	
			33.5	$P_m + P_b$	47.1		
		Outer Shell	17.7	P_m	20.0	6,496 – 6500	
			28.2	$P_m + P_b$	30.0		
26	1 Ft Side Drop, Cold Environment	Lid	33.9	$P_m + P_b$	46.7		2.10.10-44
		Body Containment	21.1	P_m	31.4	281 – 176	
			35.0	$P_m + P_b$	47.1		
		Outer Shell	16.9	P_m	20.0	6,496 – 6,500	
			27.2	$P_m + P_b$	30.0		

**These are membrane stress intensity (P_m) allowables except where the stresses are linearized through the thickness, then P_m is compared to the P_m allowable and $P_m + P_b$ is compared to the $P_m + P_b$ allowable.

Table 2.10.10-5
Maximum Component Stresses for Accident Conditions

Run No.	Description	Location	Maximum Stress Intensity (ksi)	Stress Type	Allowable Stress (ksi)	Linearized nodes	Figure
27	30 Ft. End Drop on Bottom End, Hot Environment	Lid	16.5	$P_m + P_b$	98.0		2.10.10-45
		Body Containment	24.3	$P_m + P_b$	65.94		
		Outer Shell	14.8	$P_m + P_b$	48.0		
28	30 Ft. Bottom End Drop, Cold Environment	Lid	18.9	$P_m + P_b$	98.0		2.10.10-46
		Body Containment	24.5	$P_m + P_b$	65.94		
		Outer Shell	12.5	$P_m + P_b$	48.0		
29	30 Ft. End Drop on Lid End, Hot Environment	Lid	11.9	$P_m + P_b$	98.0		2.10.10-47
		Body Containment	29.8	$P_m + P_b$	65.94		
		Outer Shell	15.7	$P_m + P_b$	48.0		
30	30 Ft. End Drop on Lid End, Cold Environment	Lid	14.3	$P_m + P_b$	98.0		2.10.10-48
		Body Containment	28.4	$P_m + P_b$	65.94		
		Outer Shell	13.5	$P_m + P_b$	48.0		
31	30 Ft. Side Drop, Hot Environment	Lid	40.7	$P_m + P_b$	98.0		2.10.10-49
		Body Containment	50.5	P_m	65.94	276 – 281	
			82.8	$P_m + P_b$	94.2		
		Outer Shell	31.2	P_m	48.0	274 – 278	
			37.9	$P_m + P_b$	72.0		
32	30 Ft. Side Drop, Cold Environment	Lid	44.8	$P_m + P_b$	98.0		2.10.10-50
		Body Containment	51.3	P_m	65.94	276 – 281	
			84.5	$P_m + P_b$	94.2		
		Outer Shell	33.0	P_m	48.0	274 – 278	
			39.8	$P_m + P_b$	72.0		
33	30 Ft. CG Over Bottom Corner Drop, Hot Environment	Lid	39.1	$P_m + P_b$	98.0		2.10.10-51
		Body Containment	61.6	P_m	65.94	15,999 – 16,152	
			92.0	$P_m + P_b$	94.2		
		Outer Shell	20.9	$P_m + P_b$	48.0		
34	30 Ft. CG Over Bottom Corner Drop, Cold Environment	Lid	31.5	$P_m + P_b$	98.0		2.10.10-52
		Body Containment	45.7	P_m	65.94	17,239 – 17,241	
			91.5	$P_m + P_b$	94.2		
		Outer Shell	19.0	$P_m + P_b$	48.0		
35	30 Ft. CG Over Lid Corner Drop, Hot Environment	Lid	6.0	P_m	98.0	13,692 – 14,046	2.10.10-53
			115.7	$P_m + P_b$	140.0		
		Body Containment	55.9	$P_m + P_b$	65.94		
		Outer Shell	20.8	$P_m + P_b$	48.0		
36	30 Ft. CG Over Lid Corner Drop, Cold Environment	Lid	7.3	P_m	98.0	17,733 – 14,125	2.10.10-54
			112.8	$P_m + P_b$	140.0		
		Body Containment	64.0	$P_m + P_b$	65.94		
		Outer Shell	17.9	$P_m + P_b$	48.0		
37	30 Ft. 20° Lid Oblique Impact, Hot Environment	Lid	23.8	P_m	98.0	18,494 – 18,486	2.10.10-55
			63.8	$P_m + P_b$	140.0		
		Body Containment	56.7	P_m	65.94	25,418 – 25,407	
			78.6	$P_m + P_b$	94.2		
		Outer Shell	45.5	P_m	48.0	2,825 – 3,065	
			59.9	$P_m + P_b$	72.0		

Table 2.10.10-5 (continued)
Maximum Component Stresses for Accident Conditions

Run No.	Description	Location	Maximum Stress Intensity (ksi)	Stress Type	Allowable Stress (ksi)		Figure
38	30 Ft. 20° Lid Oblique Impact, Cold Environment	Lid	25.7	P_m	98.0	18,494 – 18,499	2.10.10-56
			70.5	$P_m + P_b$	140.0		
		Body Containment	57.3	P_m	65.94	2,827 – 3,067	
			82.1	$P_m + P_b$	94.2		
		Outer Shell	47.8	P_m	48.0	2,825 – 3,065	
			62.6	$P_m + P_b$	72.0		
39	30 Ft. 20° Bottom Oblique Impact, Hot Environment	Lid	40.3	$P_m + P_b$	98.0		2.10.10-57
		Body Containment	46.7	P_m	65.94	7,427 – 7,422	
			72.0	$P_m + P_b$	94.2		
		Outer Shell	42.0	P_m	48.0	7,617 – 7,533	
			59.0	$P_m + P_b$	72.0		
40	30 Ft. 20° Bottom Oblique Impact, Cold Environment	Lid	30.6	$P_m + P_b$	98.0		2.10.10-58
		Body Containment	47.7	P_m	65.94	7,427 – 7,422	
			74.0	$P_m + P_b$	94.2		
		Outer Shell	43.2	P_m	48.0	7,617 – 7,533	
			60.4	$P_m + P_b$	72.0		
41	Immersion (290 psi)	Lid	17.3	$P_m + P_b$	98.0		2.10.10-59
		Body Containment	24.0	$P_m + P_b$	65.94		
		Outer Shell	9.6	$P_m + P_b$	48.0		
42	Fire Accident	Lid	24.3	$P_m + P_b$	98.0		2.10.10-60
		Body Containment	28.4	$P_m + P_b$	65.94		
		Outer Shell	39.2	$P_m + P_b$	48.0		

Figure 2.10.10-28
Lid Finite Element Plot

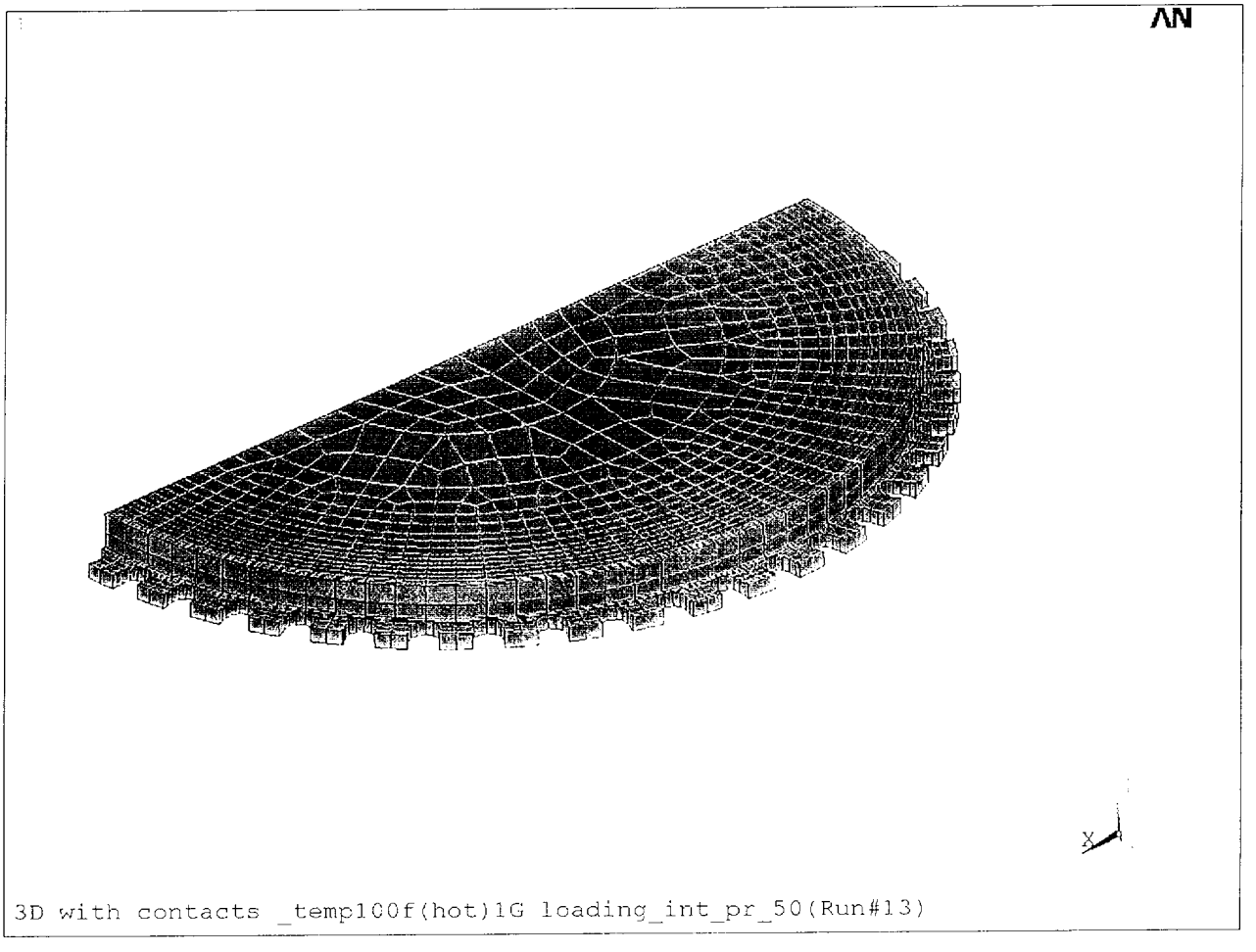


Figure 2.10.10-29
Containment Body Finite Element Plot

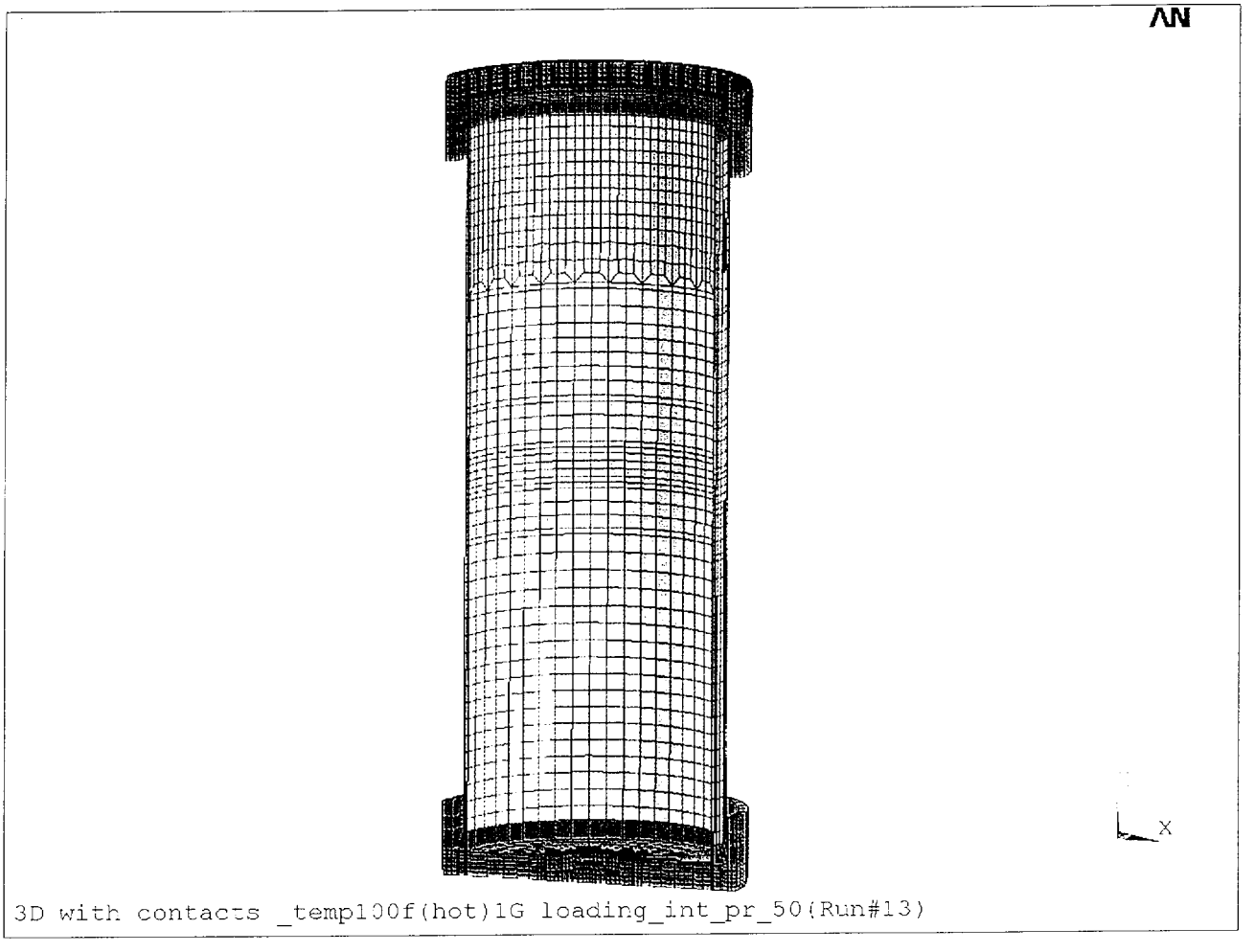


Figure 2.10.10-30
Outer Shell Finite Element Plot

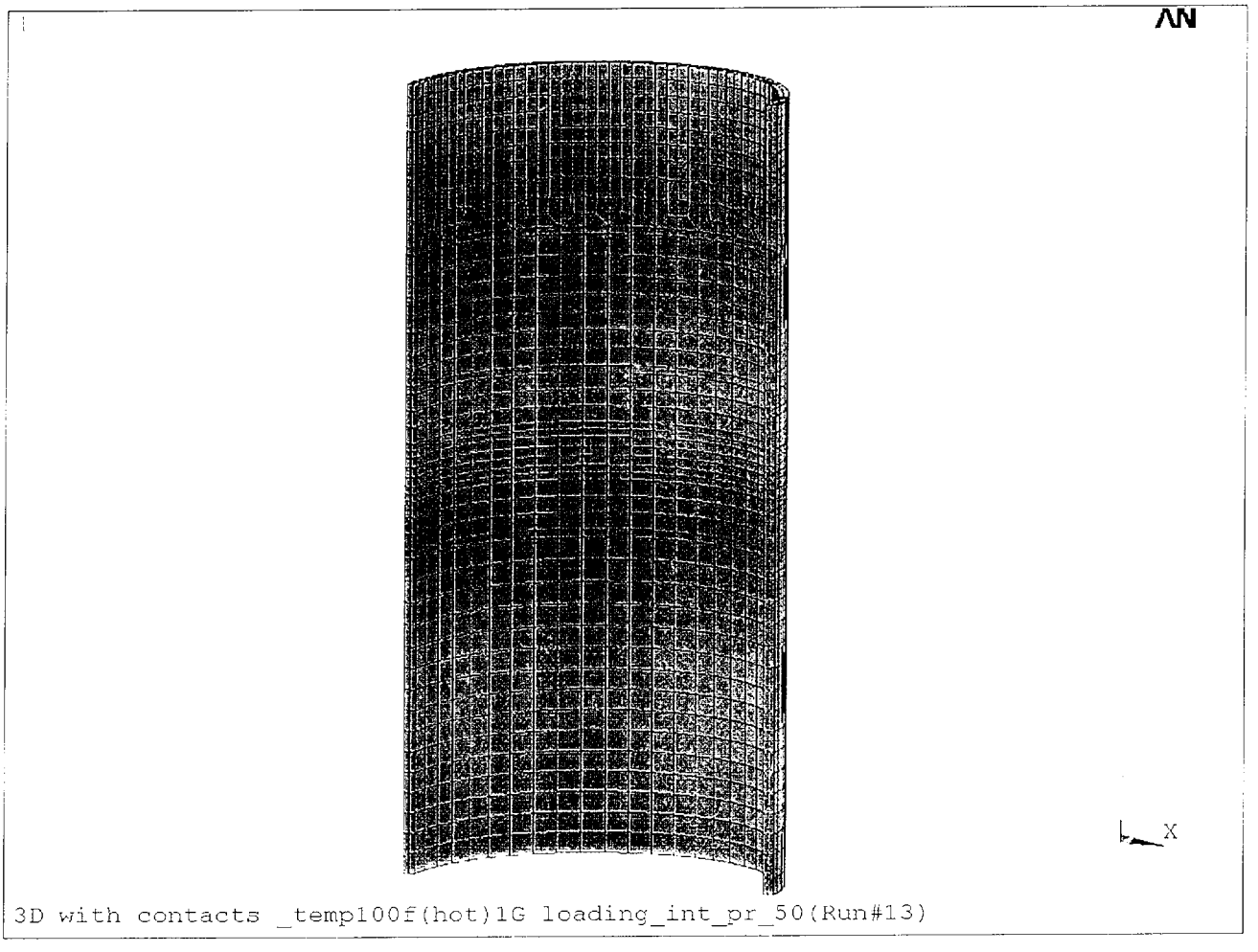


Figure 2.10.10-31
Run 13, Hot Environment (100°F ambient) Stress Intensity Plot

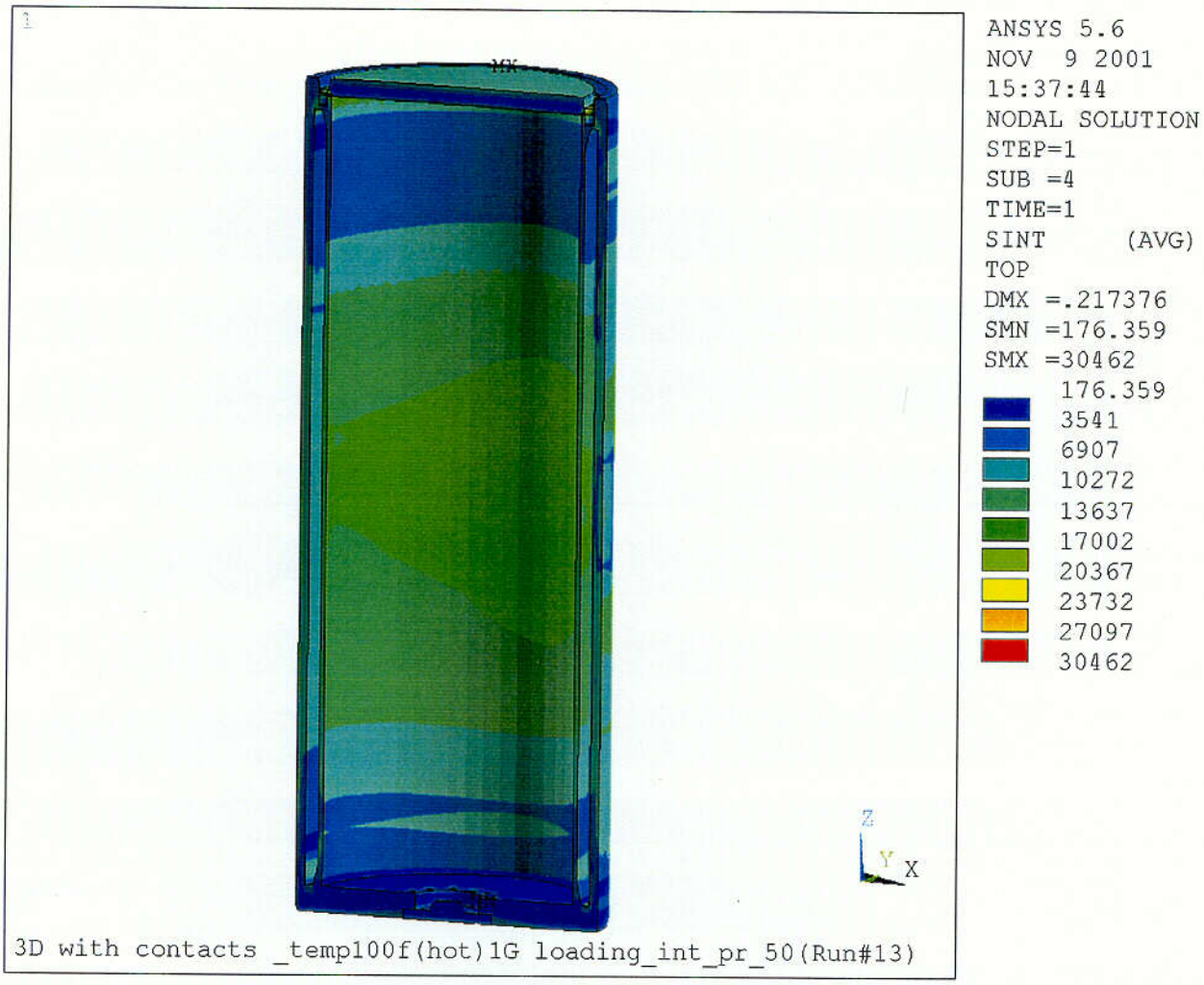


Figure 2.10.10-32
Run 14, Cold Environment (-40°F ambient) Stress Intensity Plot

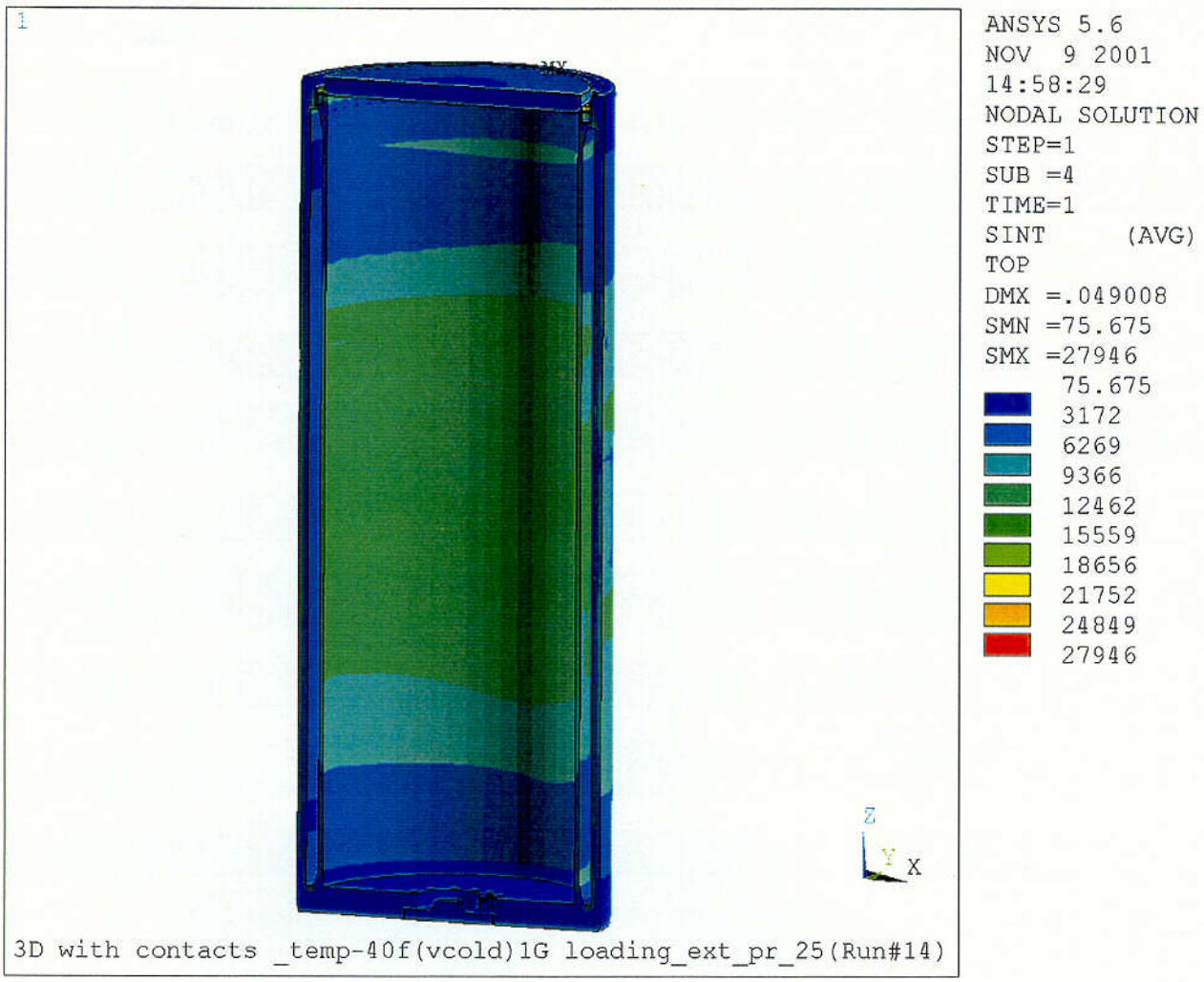


Figure 2.10.10-33
Run 15, Increased External Pressure Stress Intensity Plot

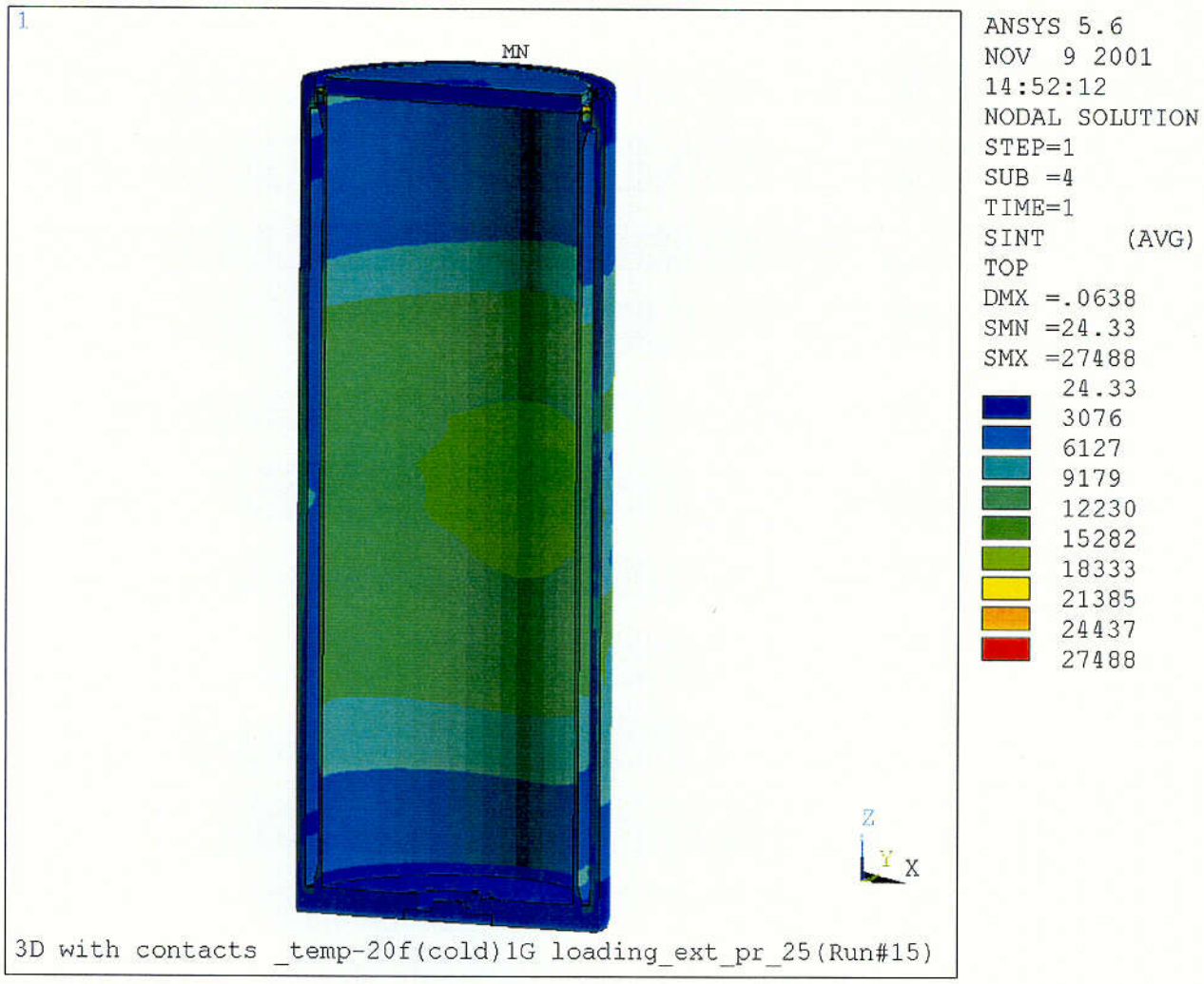


Figure 2.10.10-34
Run 16, Minimum External Pressure Stress Intensity Plot

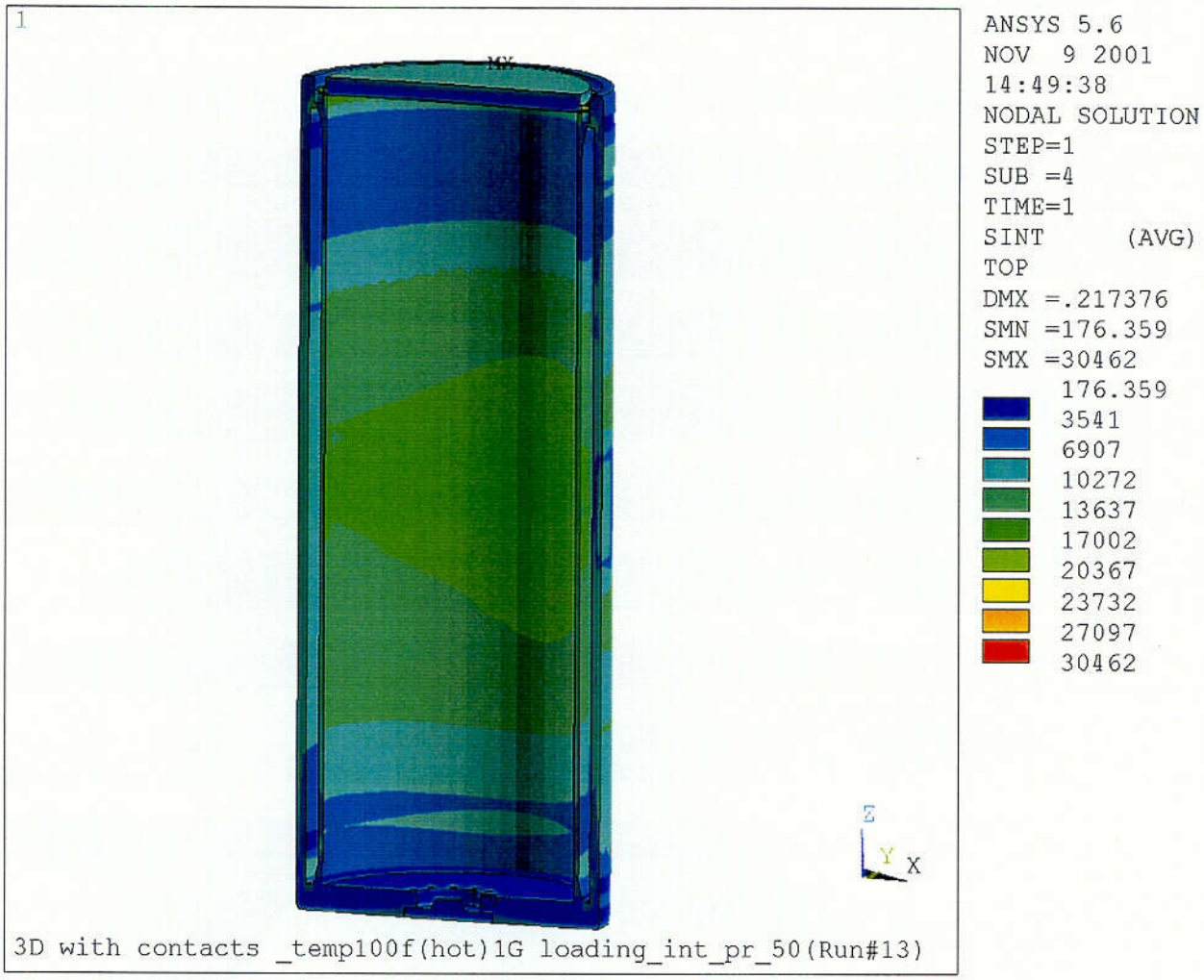


Figure 2.10.10-35
Run 17, Rail Car Vibration, Hot Environment Stress Intensity Plot

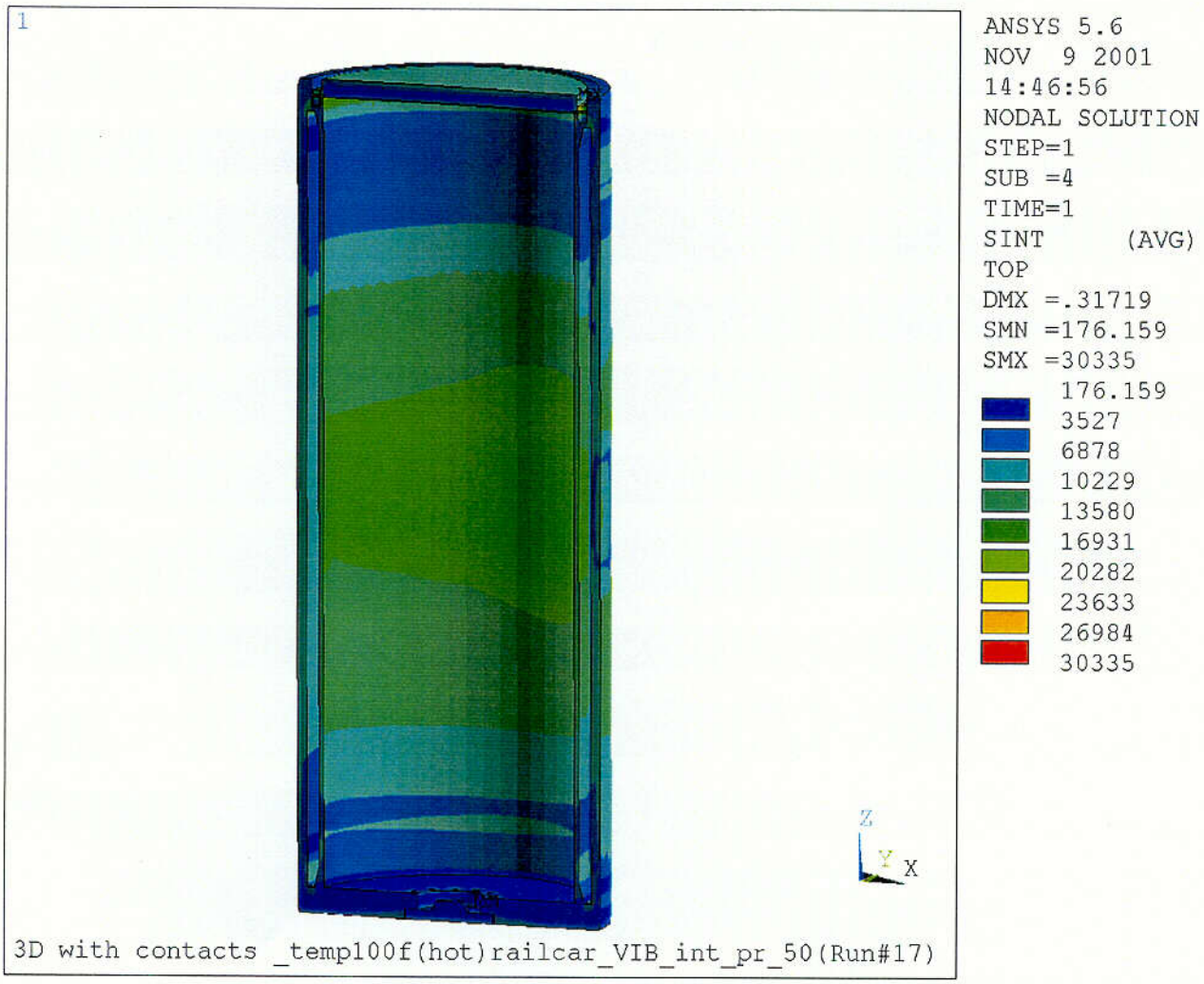


Figure 2.10.10-36
Run 18, Rail Car Vibration, Cold Environment Stress Intensity Plot

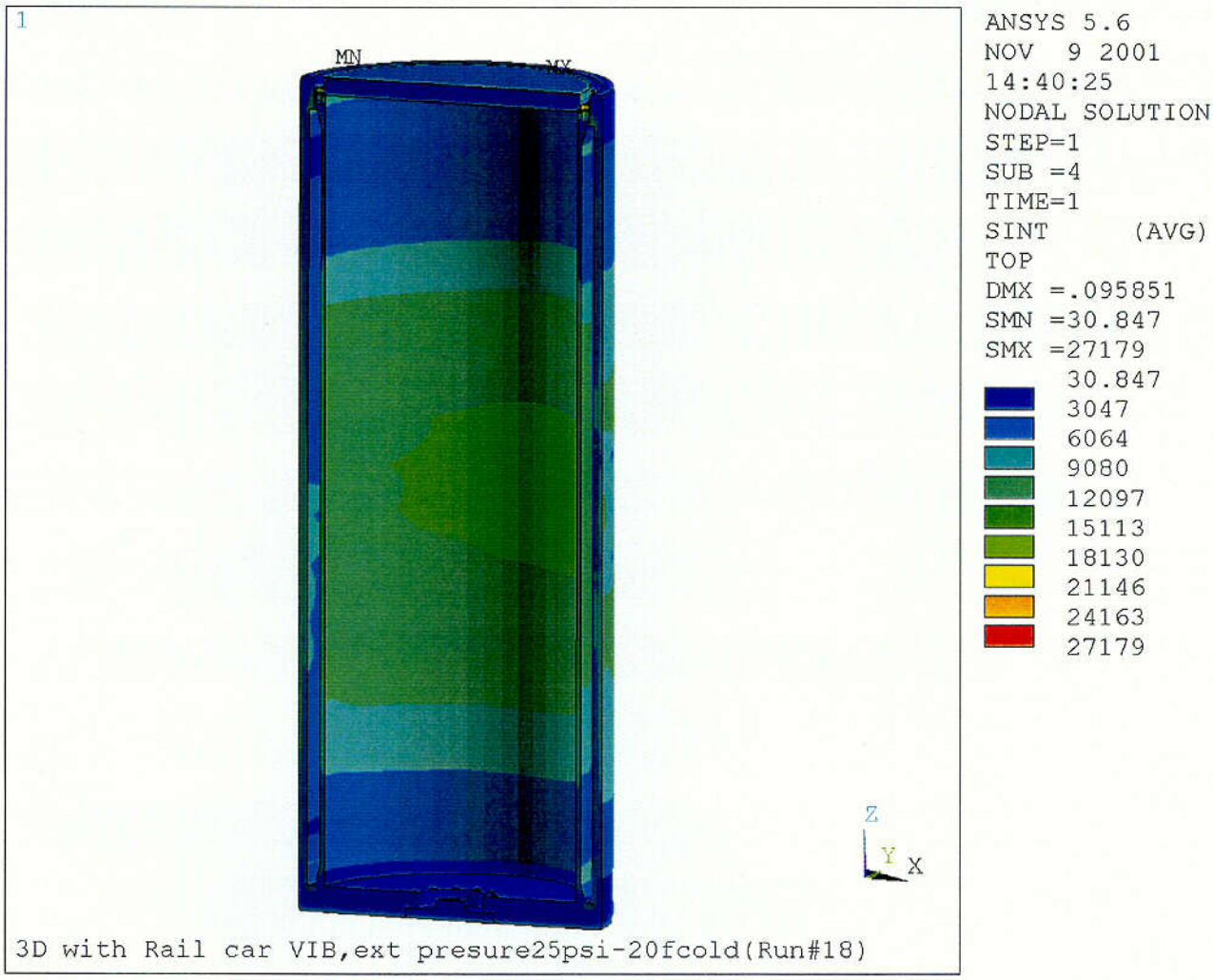


Figure 2.10.10-37
Run 19, Rail Car Shock, Hot Environment Stress Intensity Plot

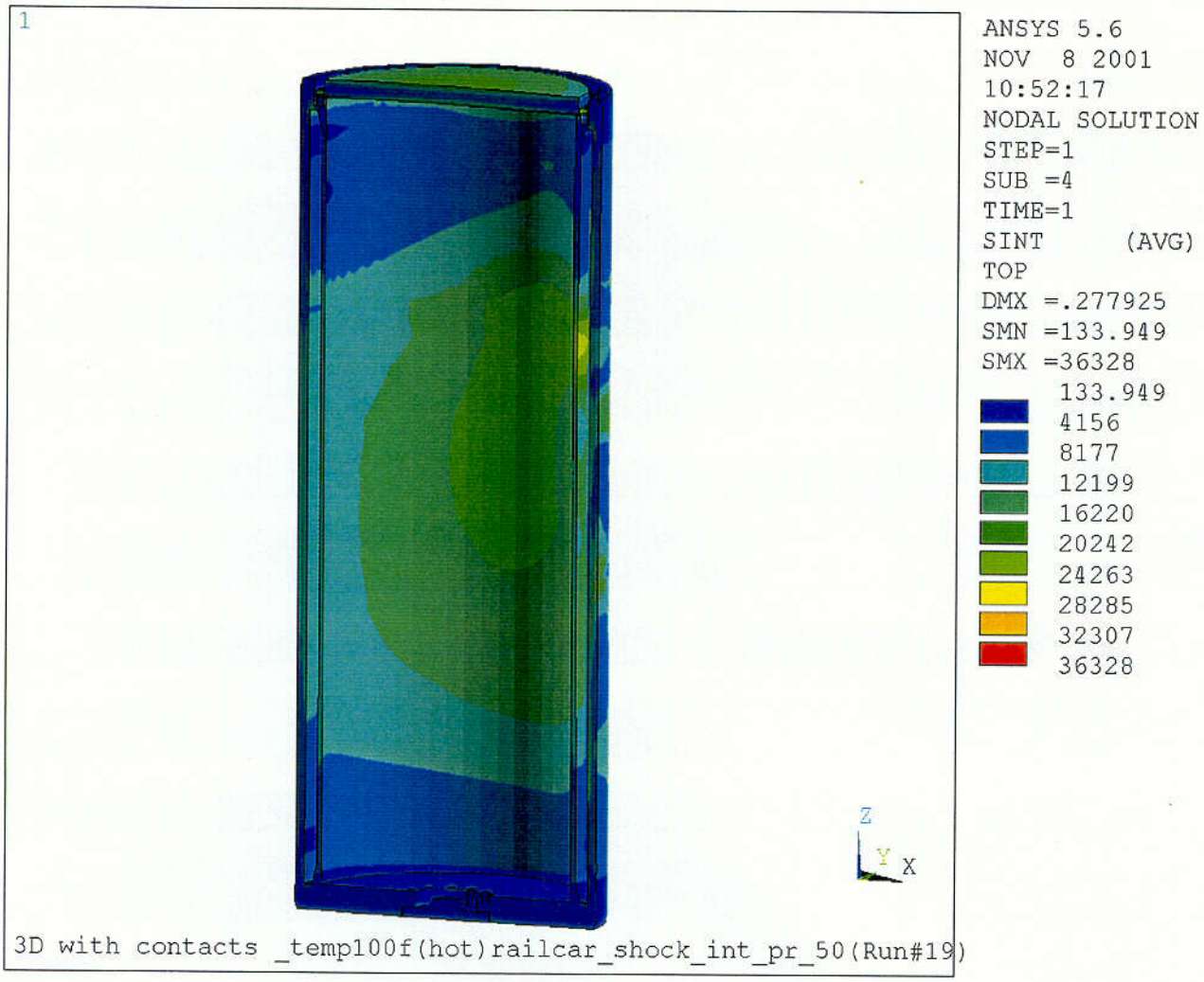


Figure 2.10.10-38
Run 20, Rail Car Shock, Cold Environment Stress Intensity Plot

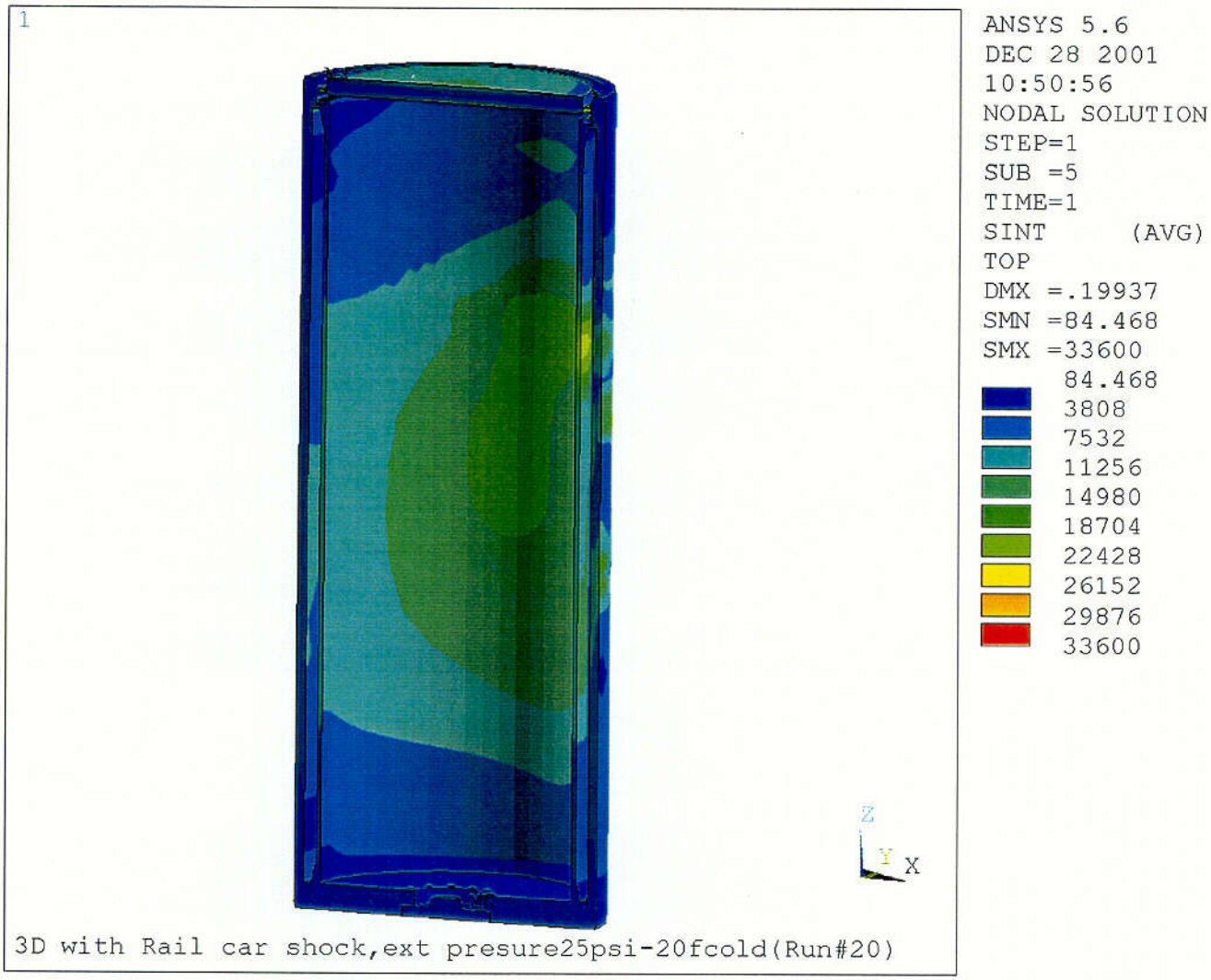


Figure 2.10.10-39
Run 21, 1 Foot Lid End Drop, Hot Environment Stress Intensity Plot

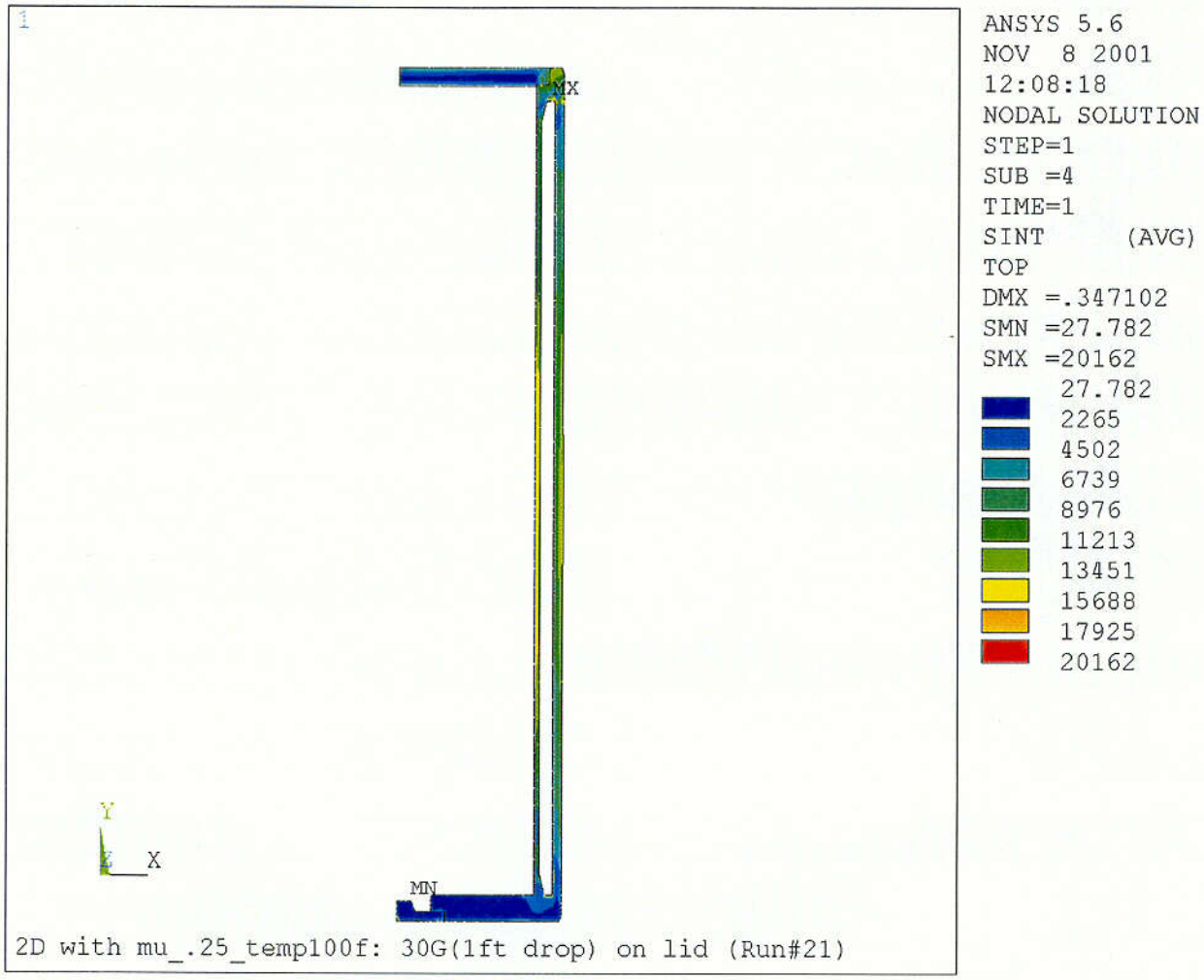


Figure 2.10.10-40
Run 22, 1 Foot Lid End Drop, Cold Environment Stress Intensity Plot

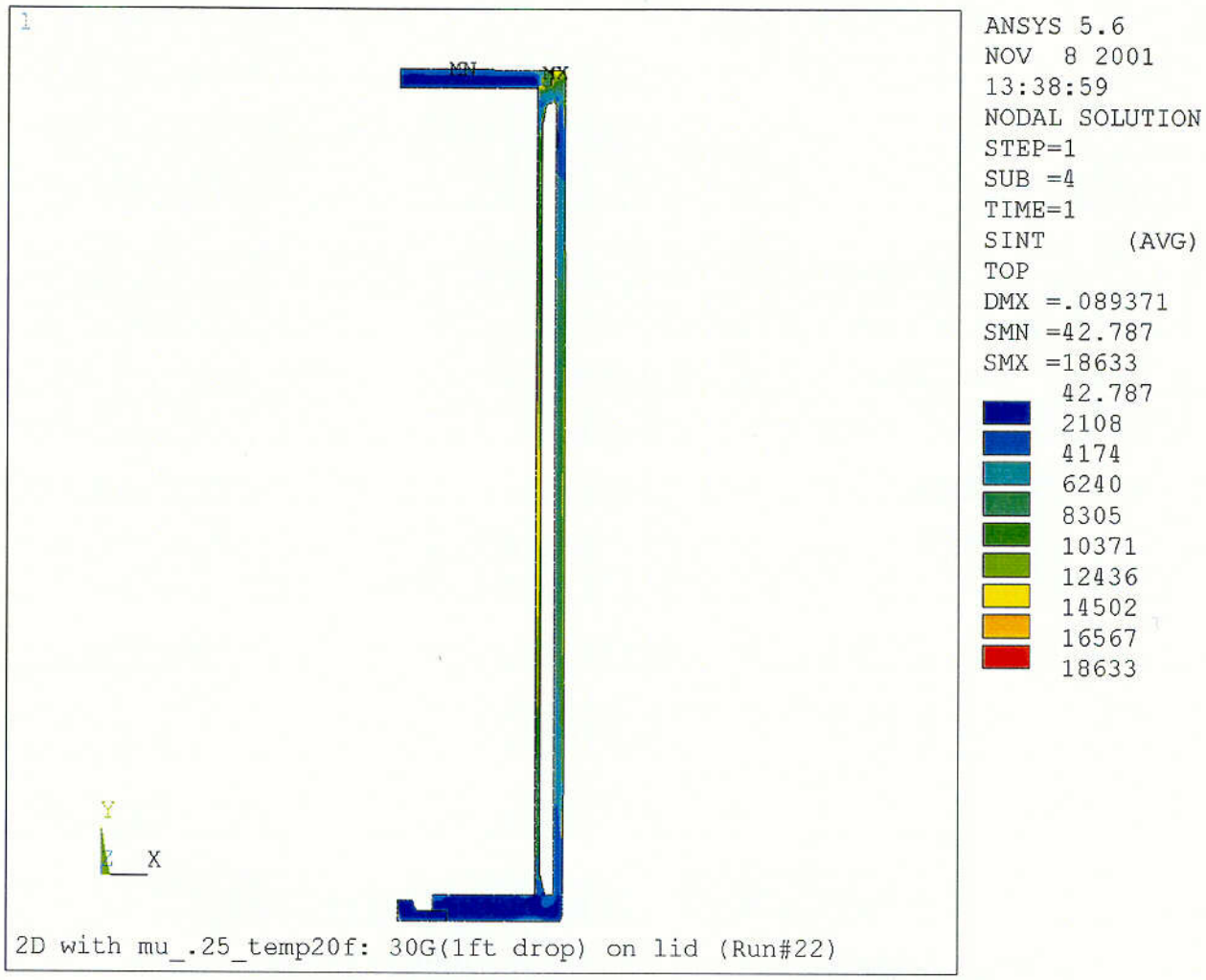


Figure 2.10.10-41
Run 23, 1 Foot Bottom End Drop, Hot Environment Stress Intensity Plot

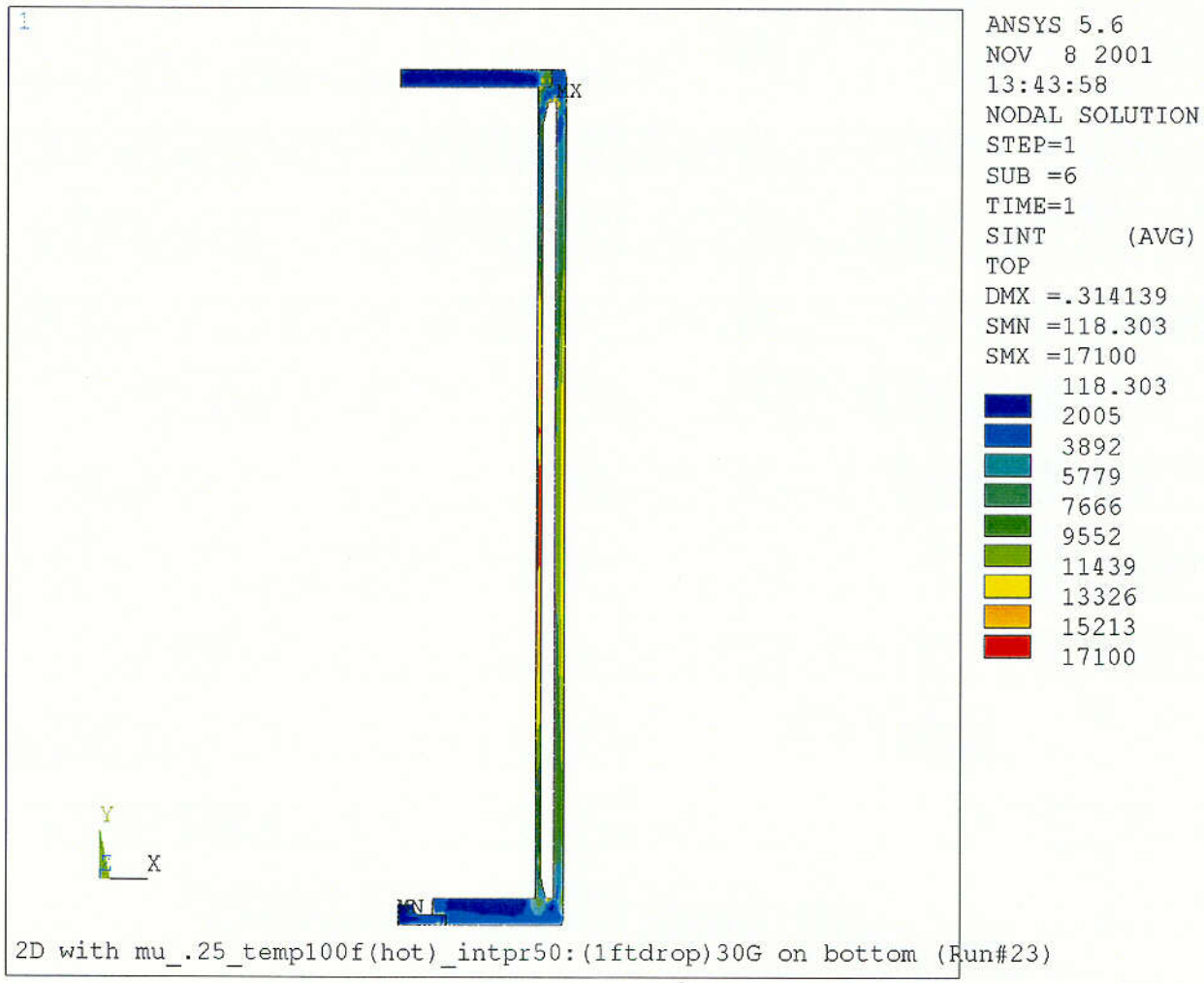


Figure 2.10.10-42
Run 24, 1 Foot Bottom End Drop, Cold Environment Stress Intensity Plot

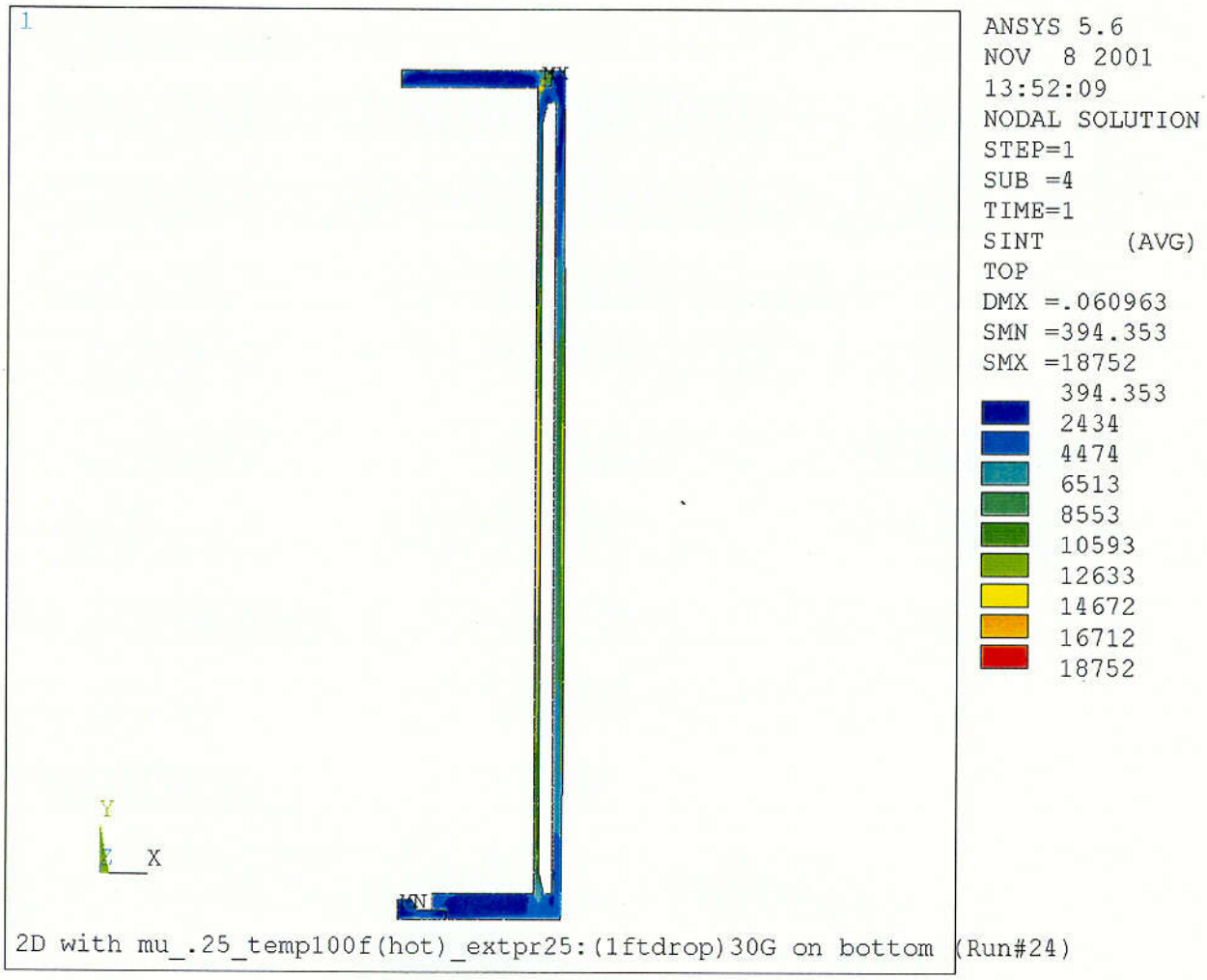


Figure 2.10.10-43
Run 25, 1 Foot Side Drop, Hot Environment Stress Intensity Plot

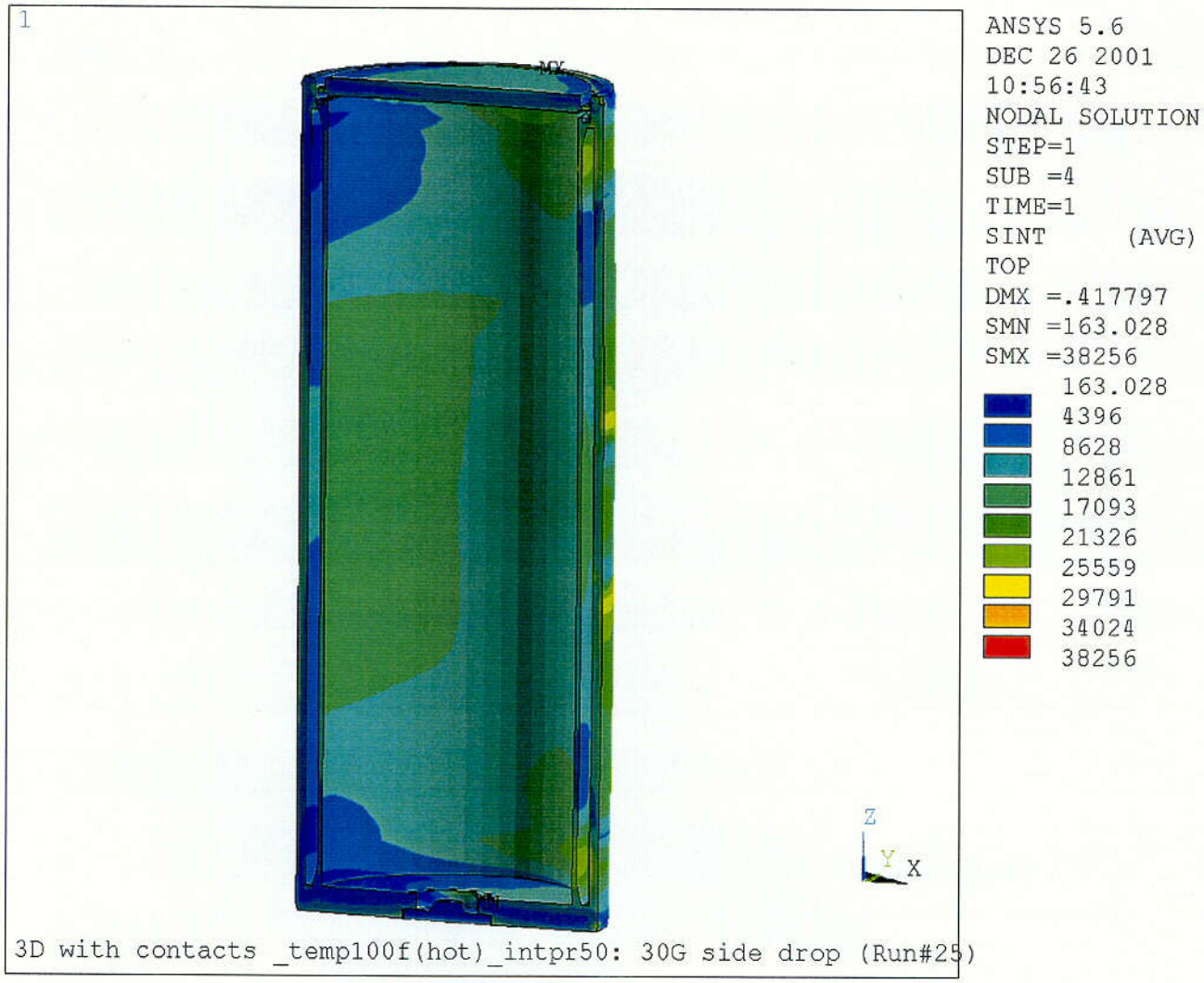


Figure 2.10.10-44
Run 26, 1 Foot Side Drop, Cold Environment Stress Intensity Plot

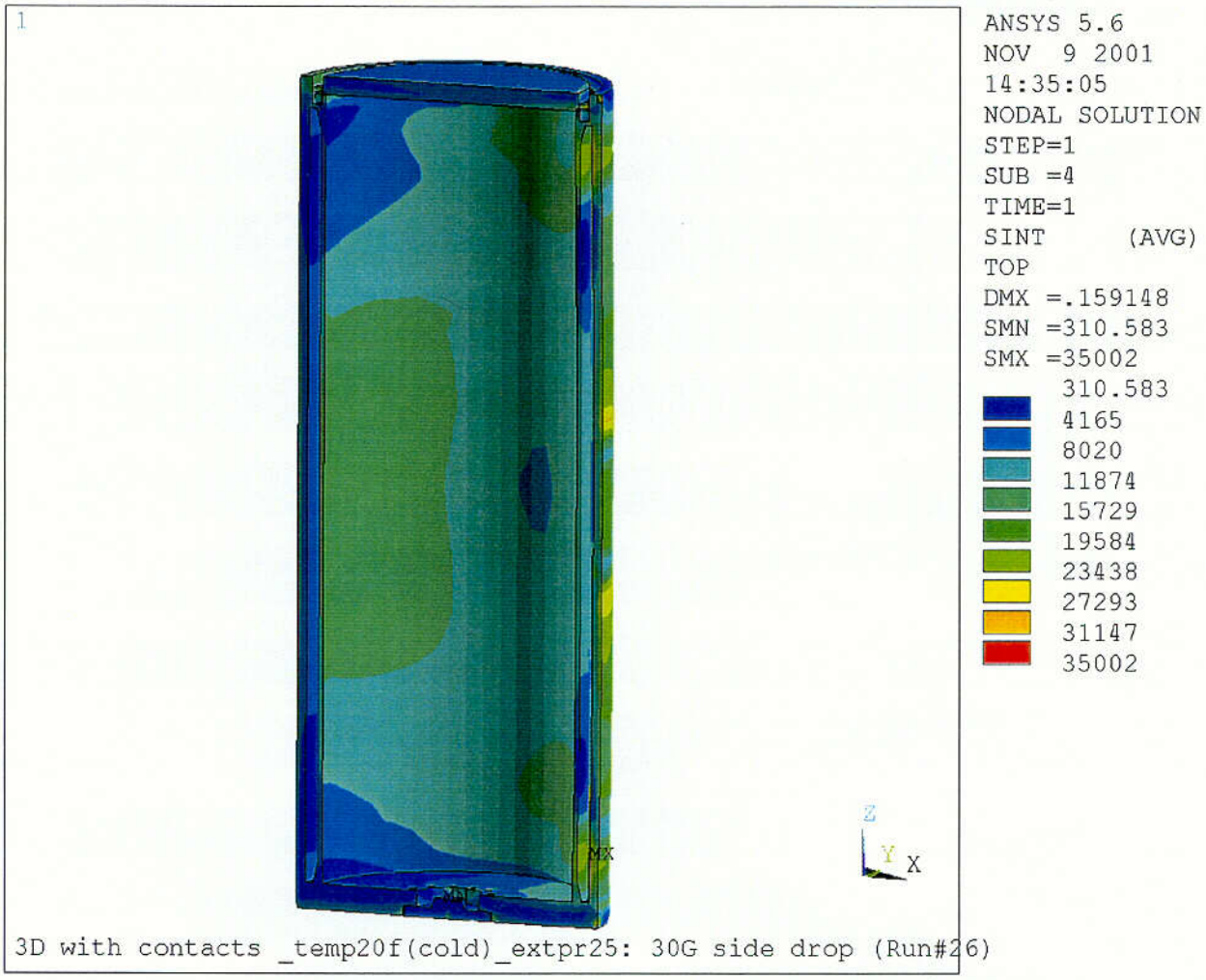


Figure 2.10.10-45
Run 27, 30 Foot Bottom End Drop, Hot Environment Stress Intensity Plot

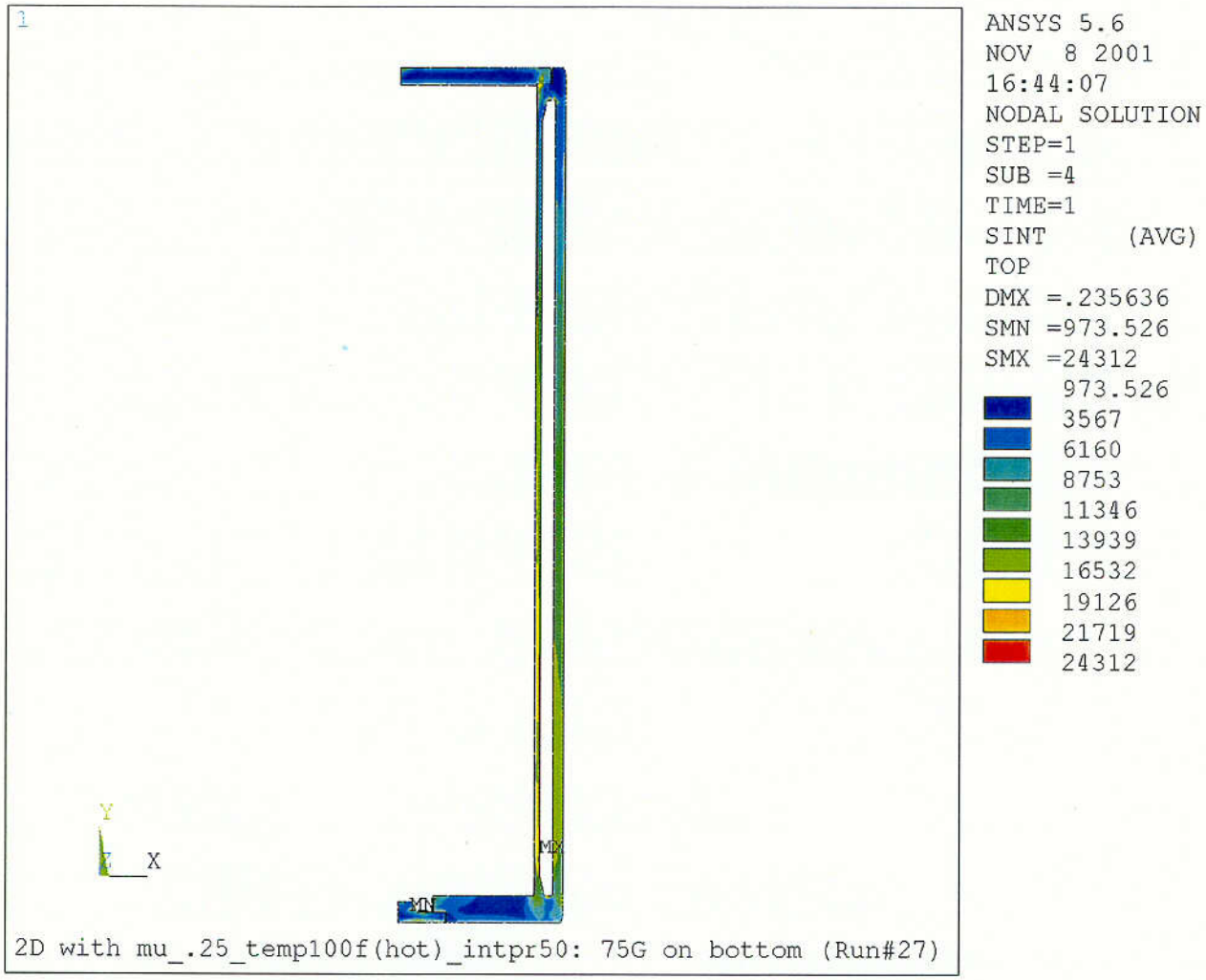


Figure 2.10.10-46
Run 28, 30 Foot Bottom End Drop, Cold Environment Stress Intensity Plot

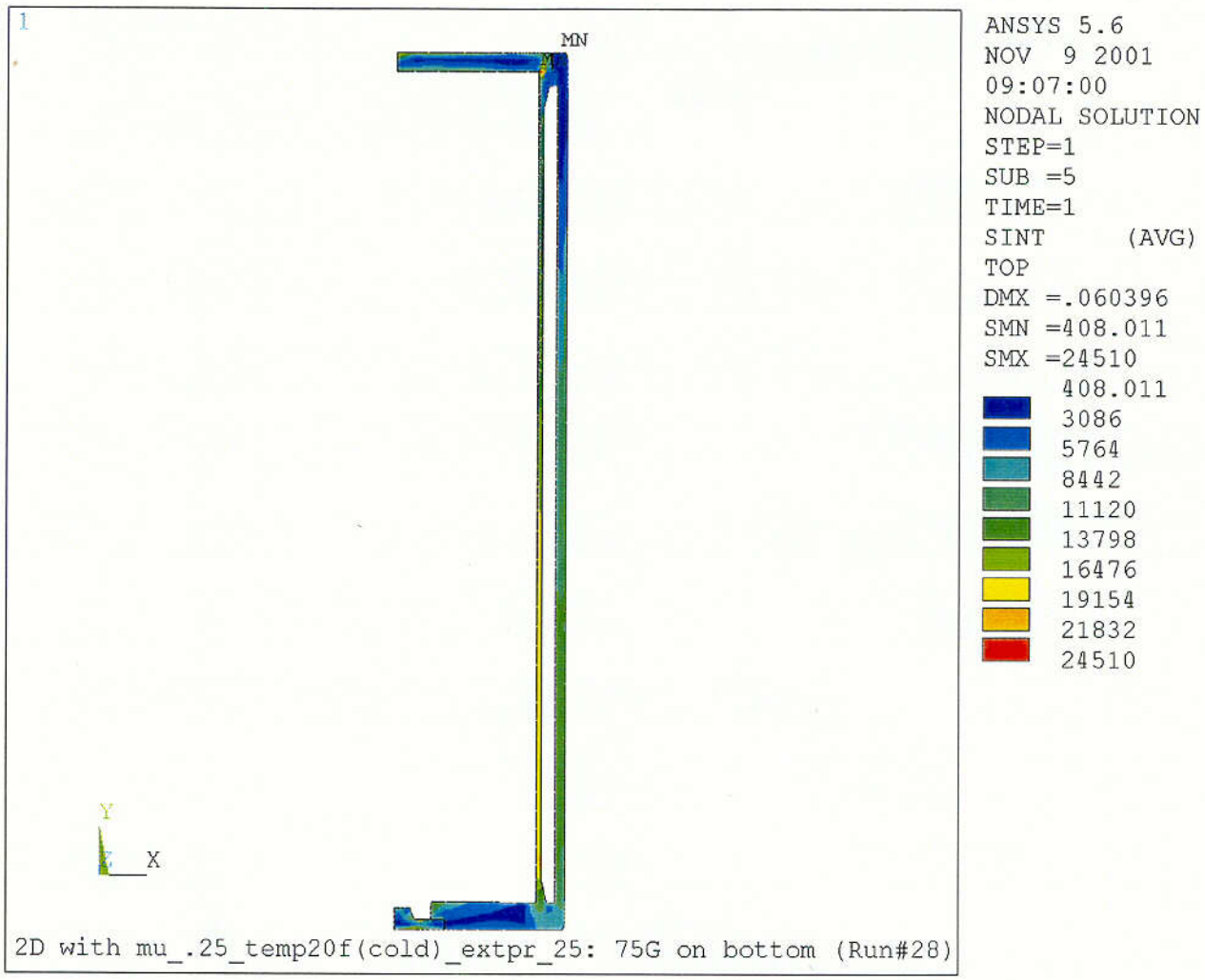


Figure 2.10.10-47
Run 29, 30 Foot Lid End Drop, Hot Environment Stress Intensity Plot

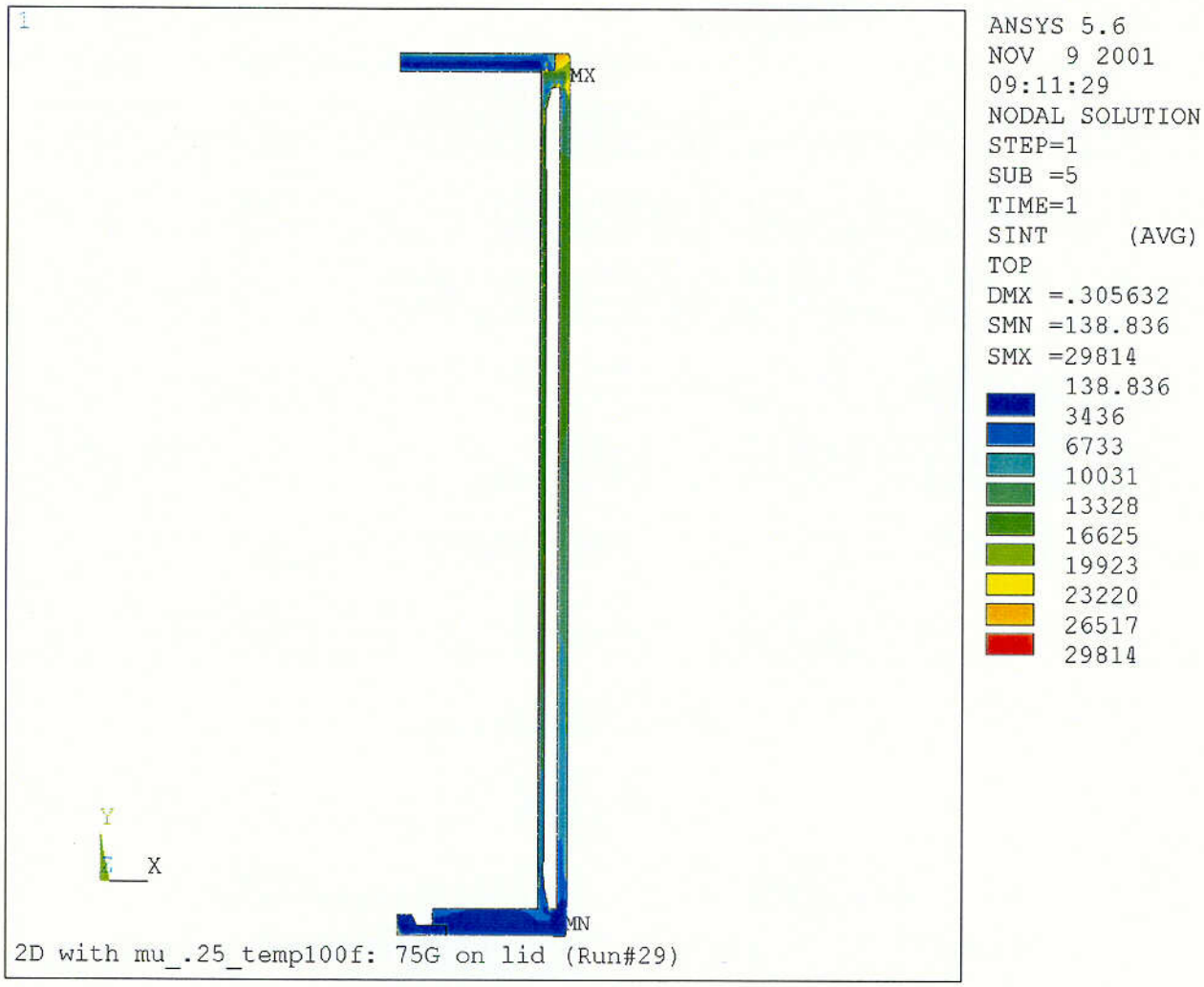


Figure 2.10.10-48
Run 30, 30 Foot Lid End Drop, Cold Environment Stress Intensity Plot

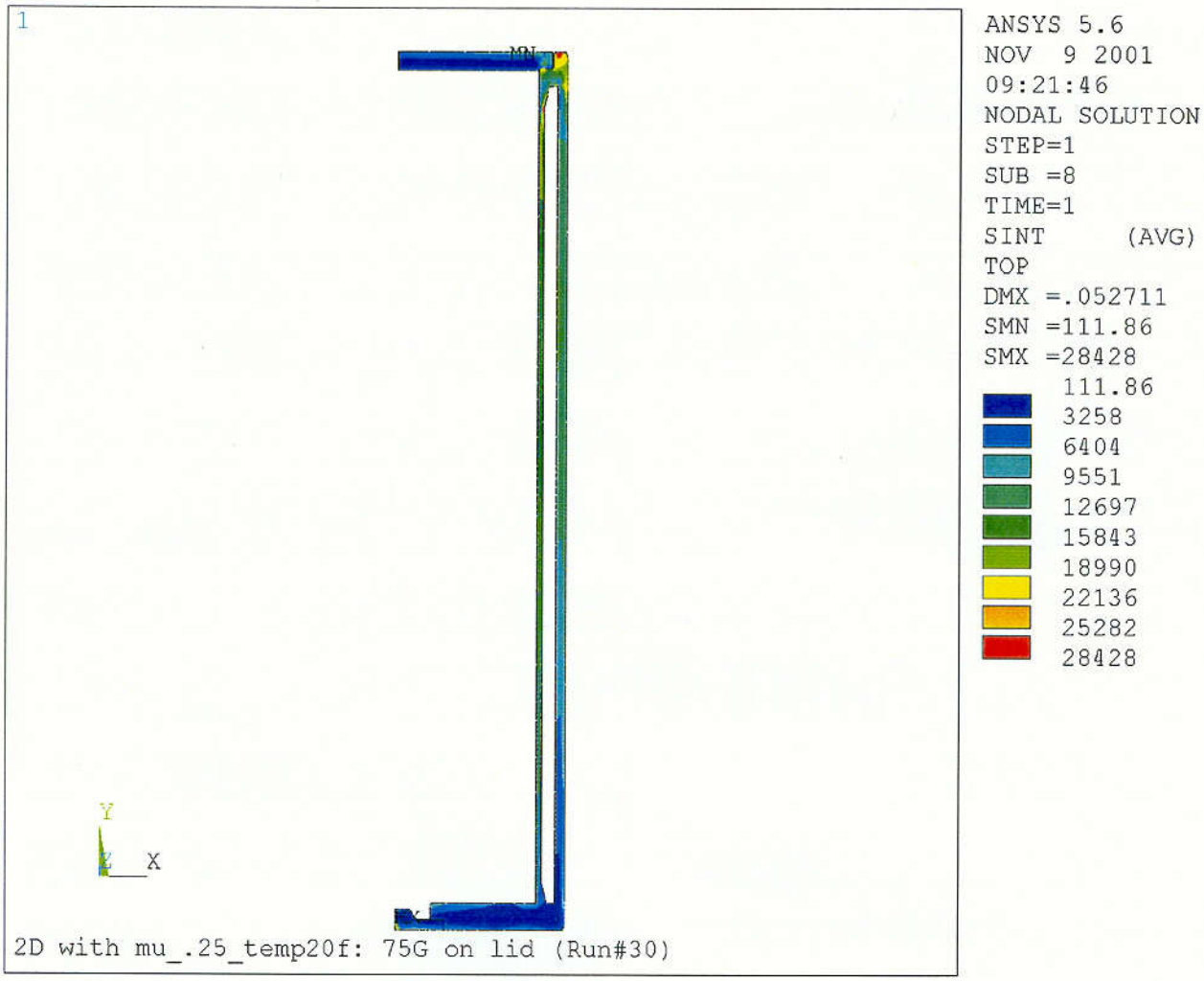


Figure 2.10.10-49
Run 31, 30 Foot Side Drop, Hot Environment Stress Intensity Plot

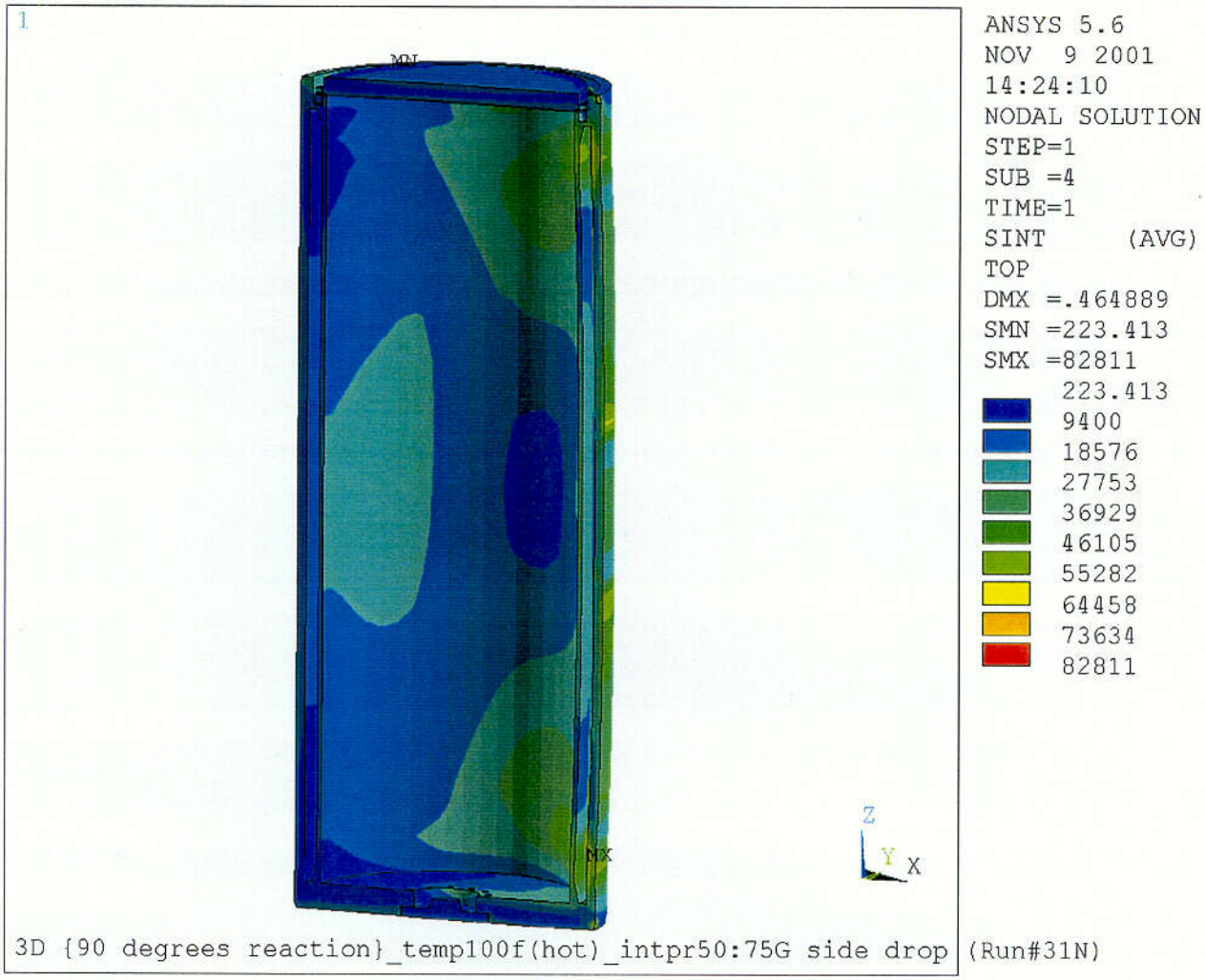


Figure 2.10.10-50
Run 32, 30 Foot Side Drop, Cold Environment Stress Intensity Plot

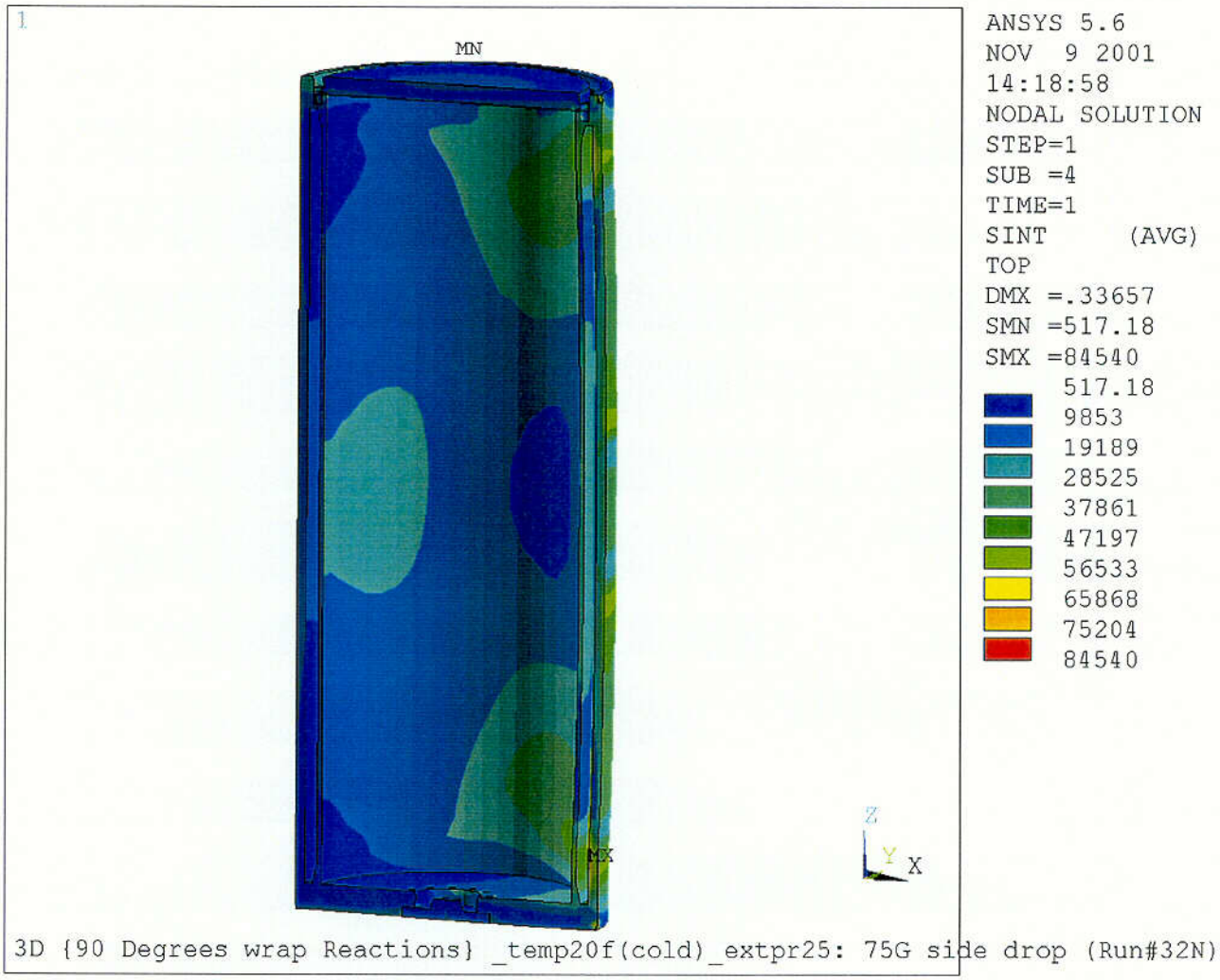


Figure 2.10.10-51
Run 33, 30 Foot CG Over Bottom Corner Drop, Hot Environment Stress Intensity Plot

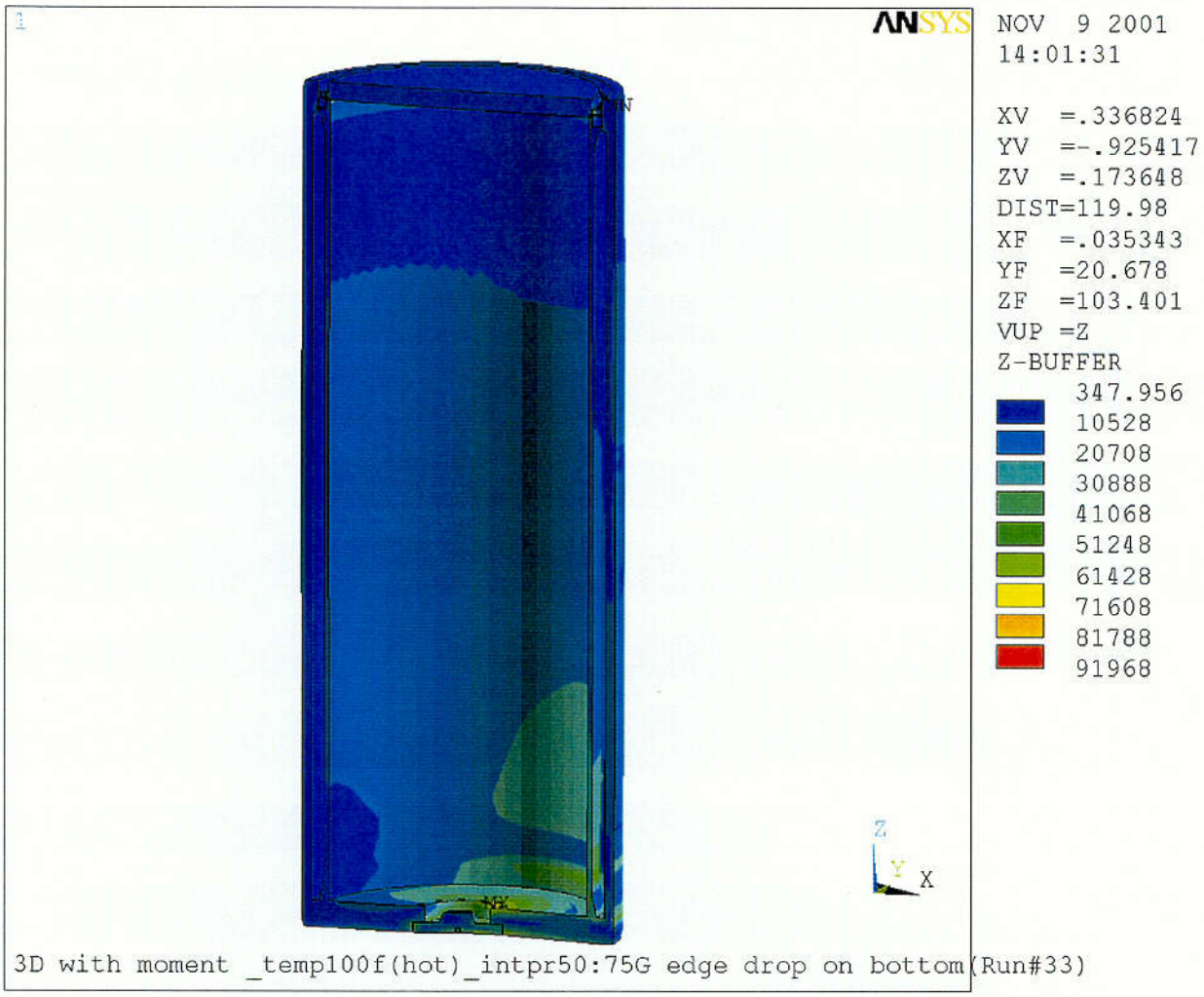


Figure 2.10.10-52
Run 34, 30 Foot CG Over Bottom Corner Drop, Cold Environment Stress Intensity Plot

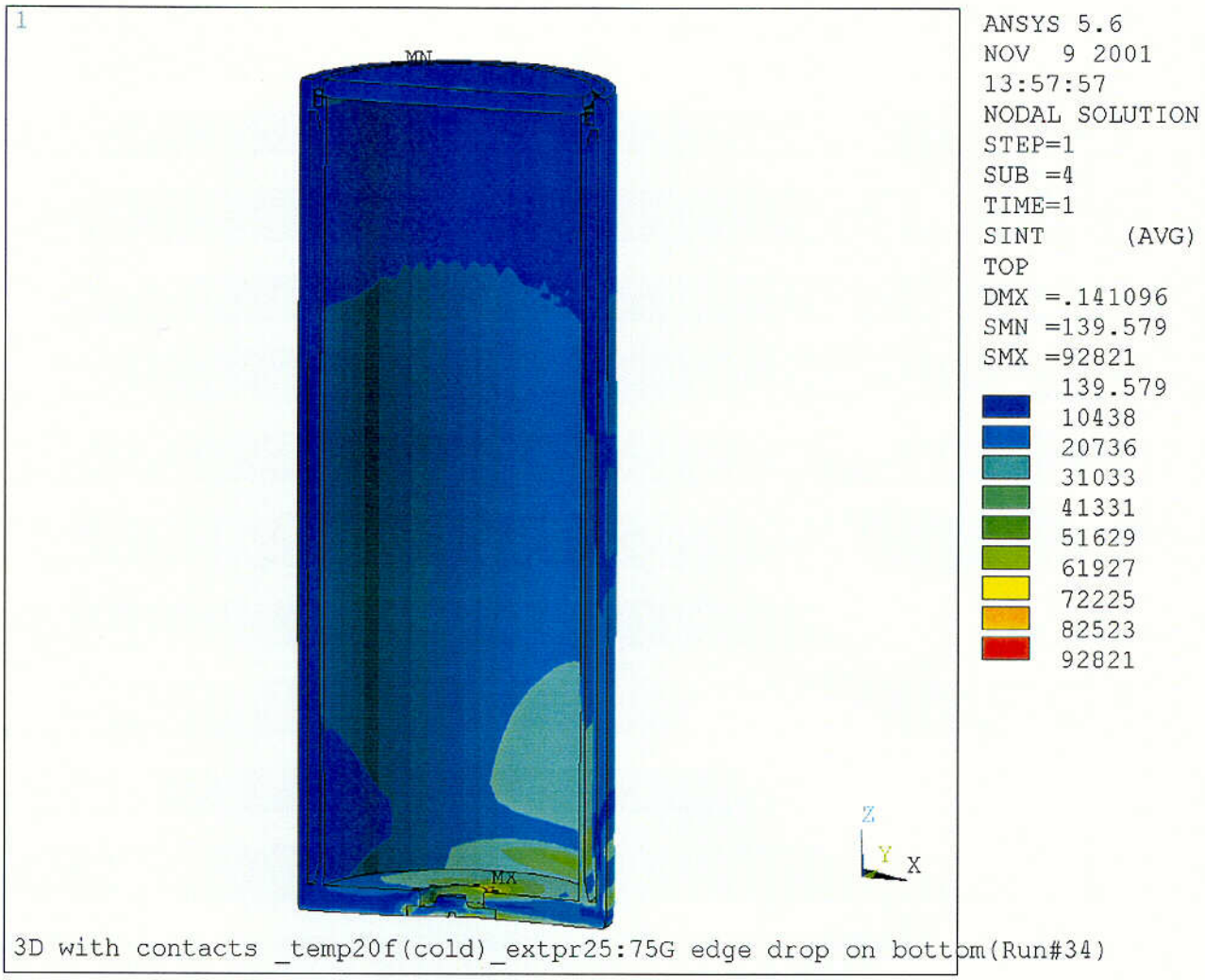


Figure 2.10.10-53
Run 35, 30 Foot CG Over Lid Corner Drop, Hot Environment Stress Intensity Plot

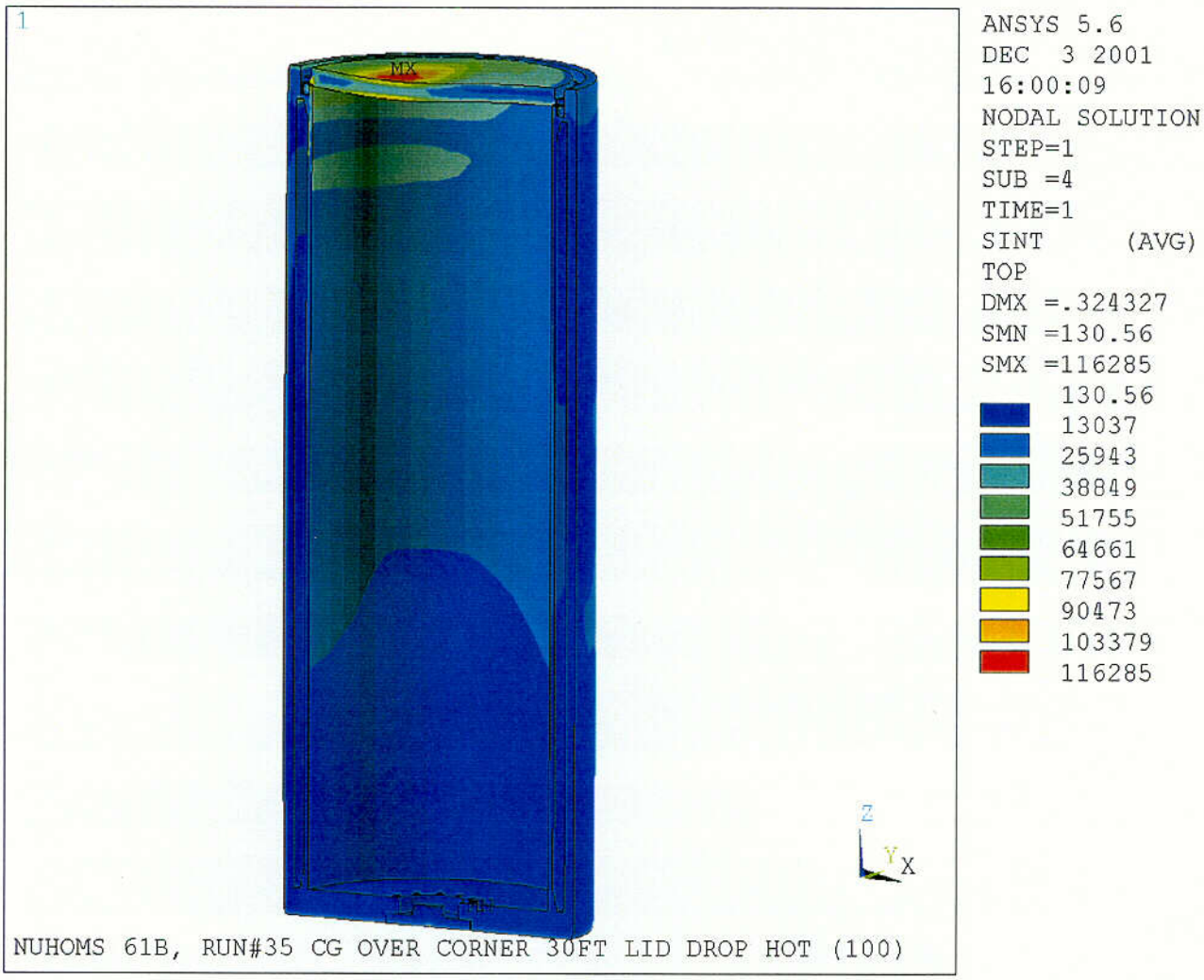


Figure 2.10.10-54
Run 36, 30 Foot CG Over Lid Corner Drop, Cold Environment Stress Intensity Plot

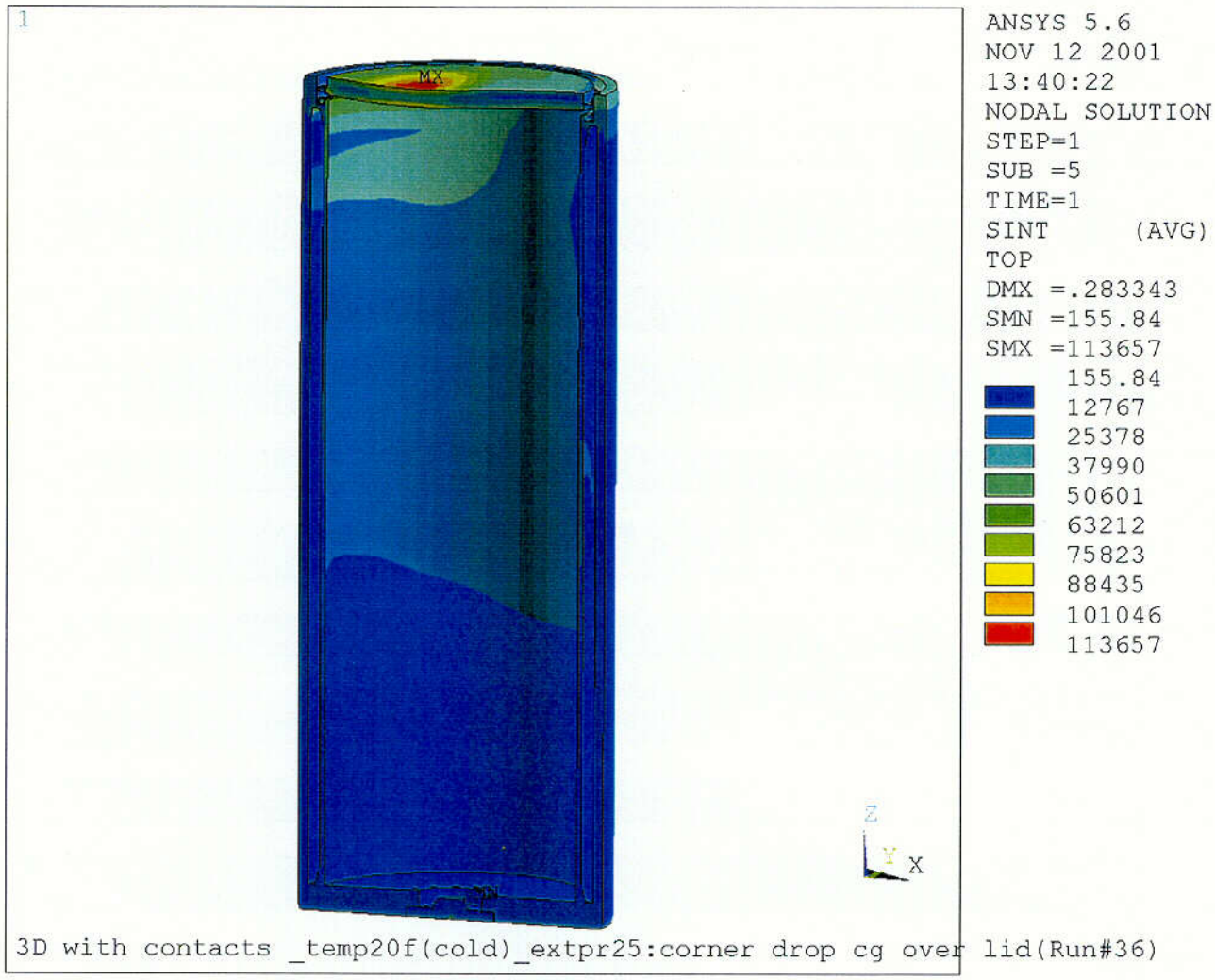


Figure 2.10.10-55
Run 37, 30 Foot 20° Lid End Slap Down Drop, Hot Environment Stress Intensity Plot

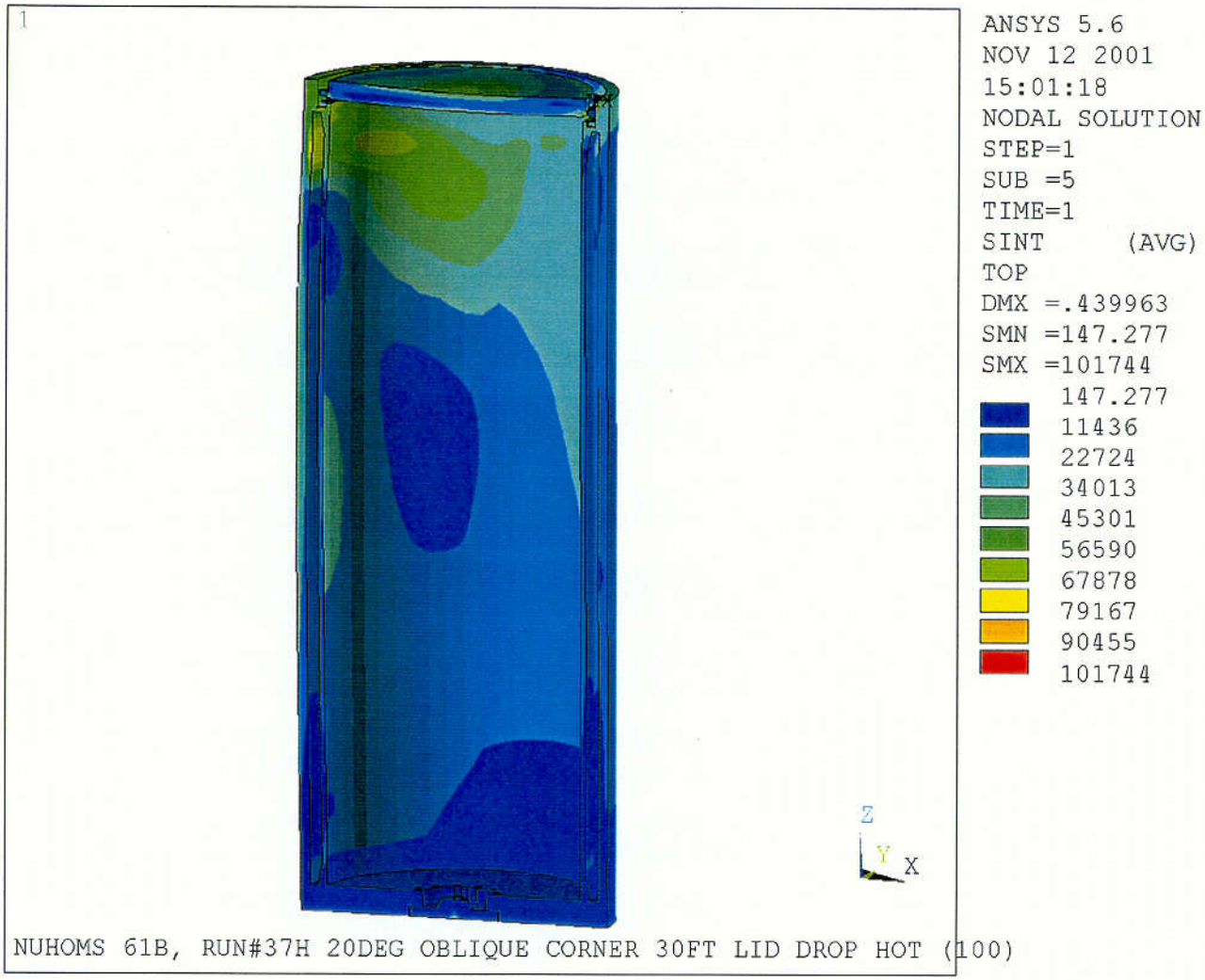


Figure 2.10.10-56
Run 38, 30 Foot 20° Lid End Slap Down Drop, Cold Environment Stress Intensity Plot

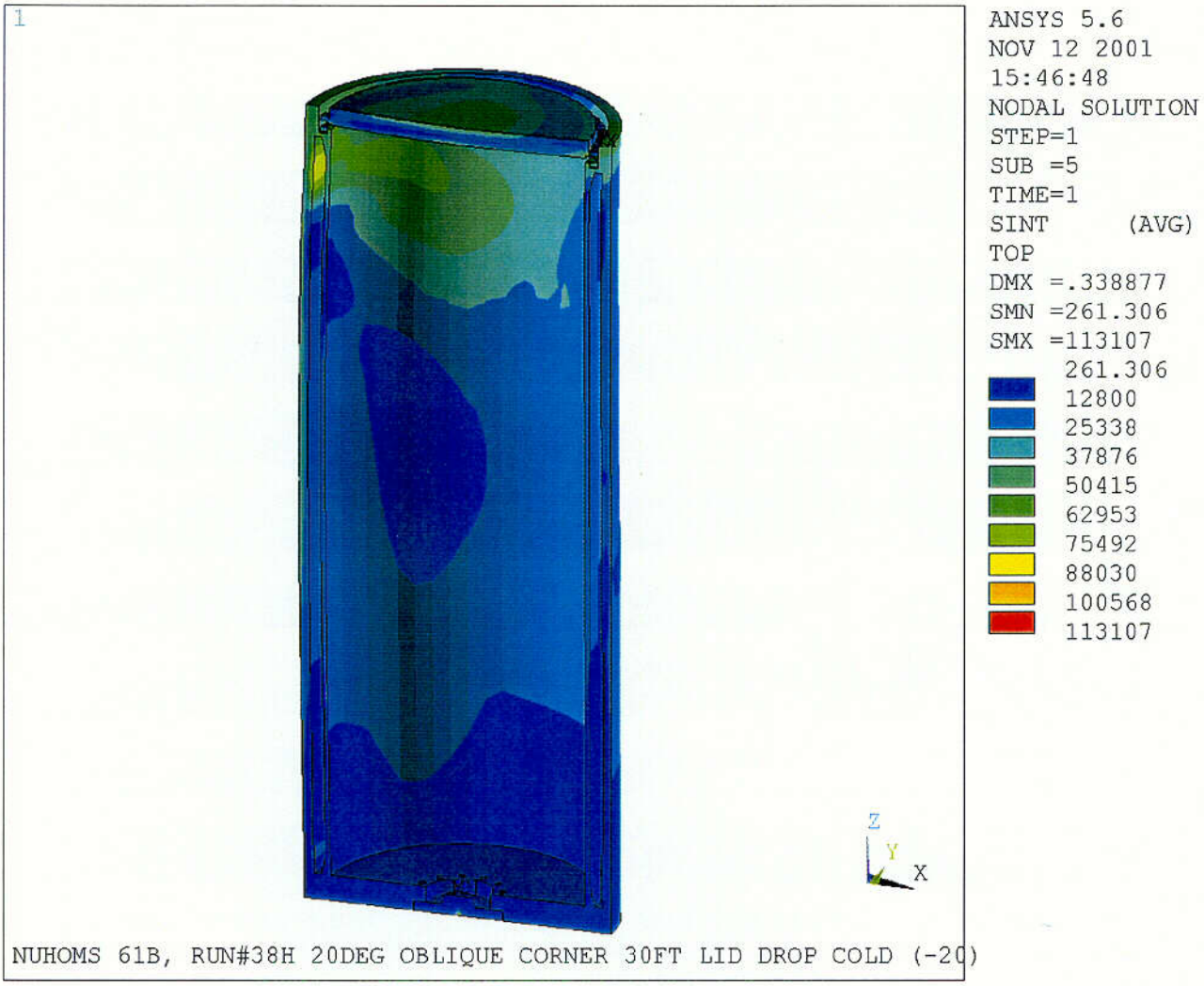


Figure 2.10.10-57
Run 39, 30 Foot 20° Bottom End Slap Down Drop, Hot Environment Stress Intensity Plot

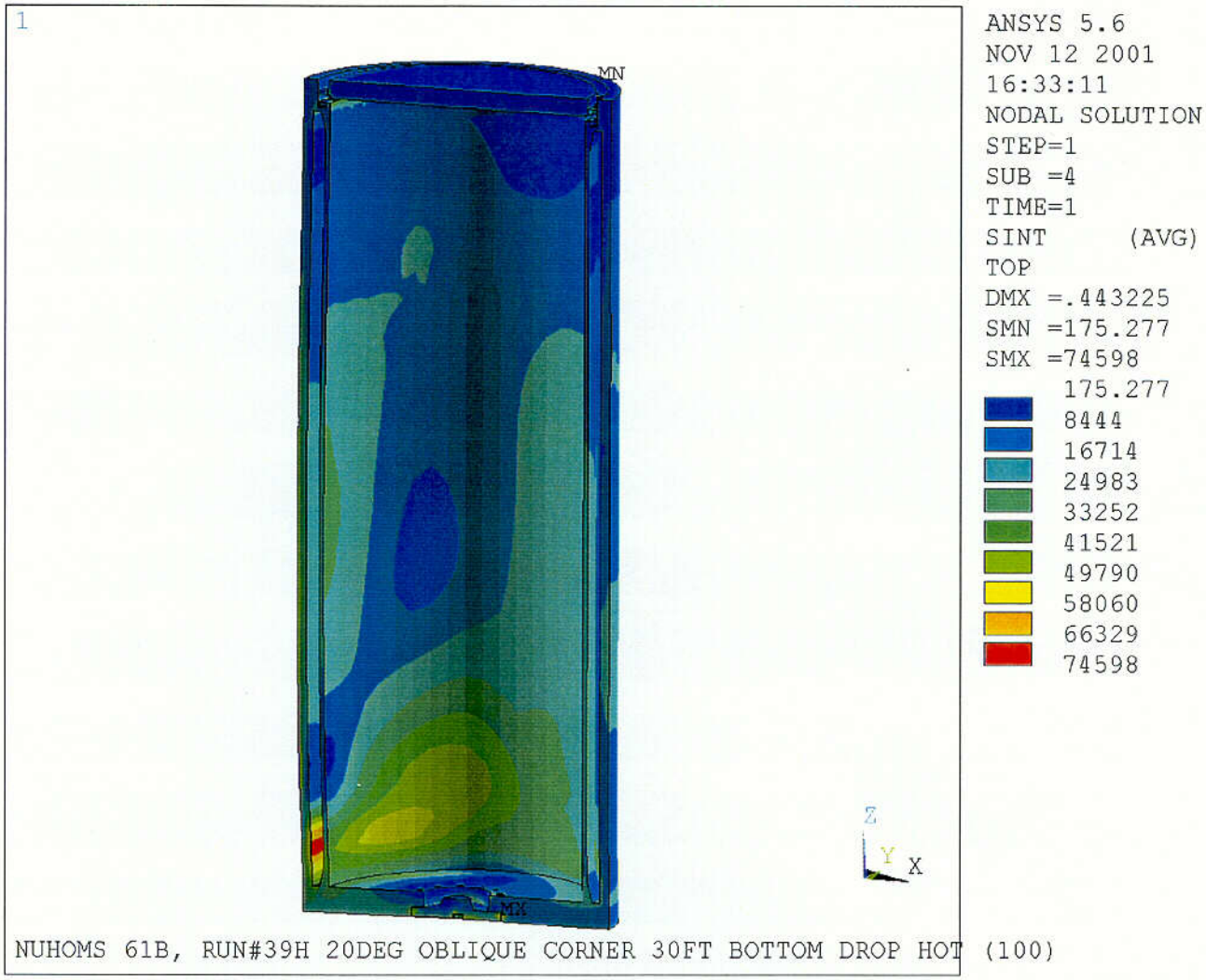


Figure 2.10.10-58

Run 40, 30 Foot 20° Bottom End Slap Down Drop, Cold Environment Stress Intensity Plot

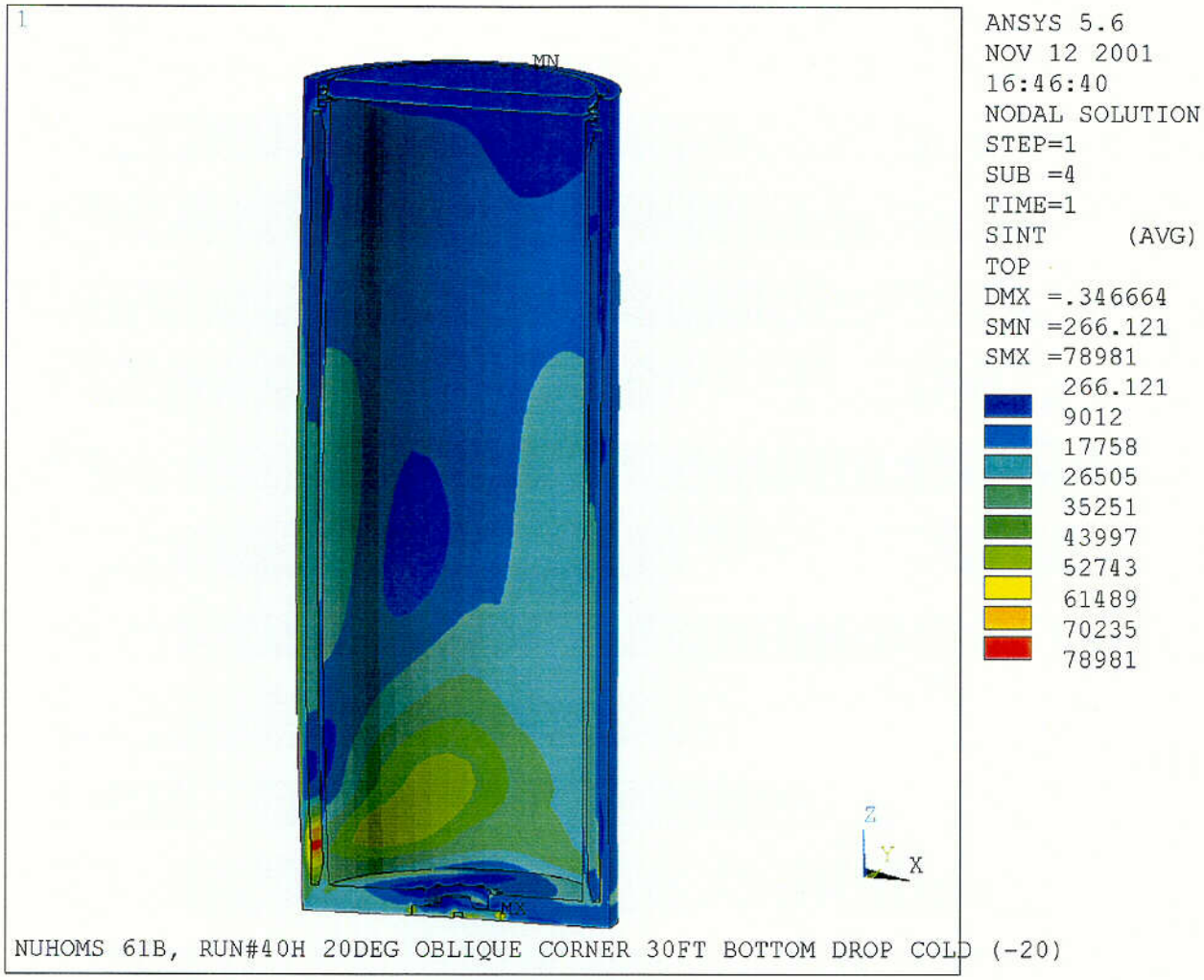


Figure 2.10.10-59
Run 41, Immersion (290 psi), Hot Environment Stress Intensity Plot

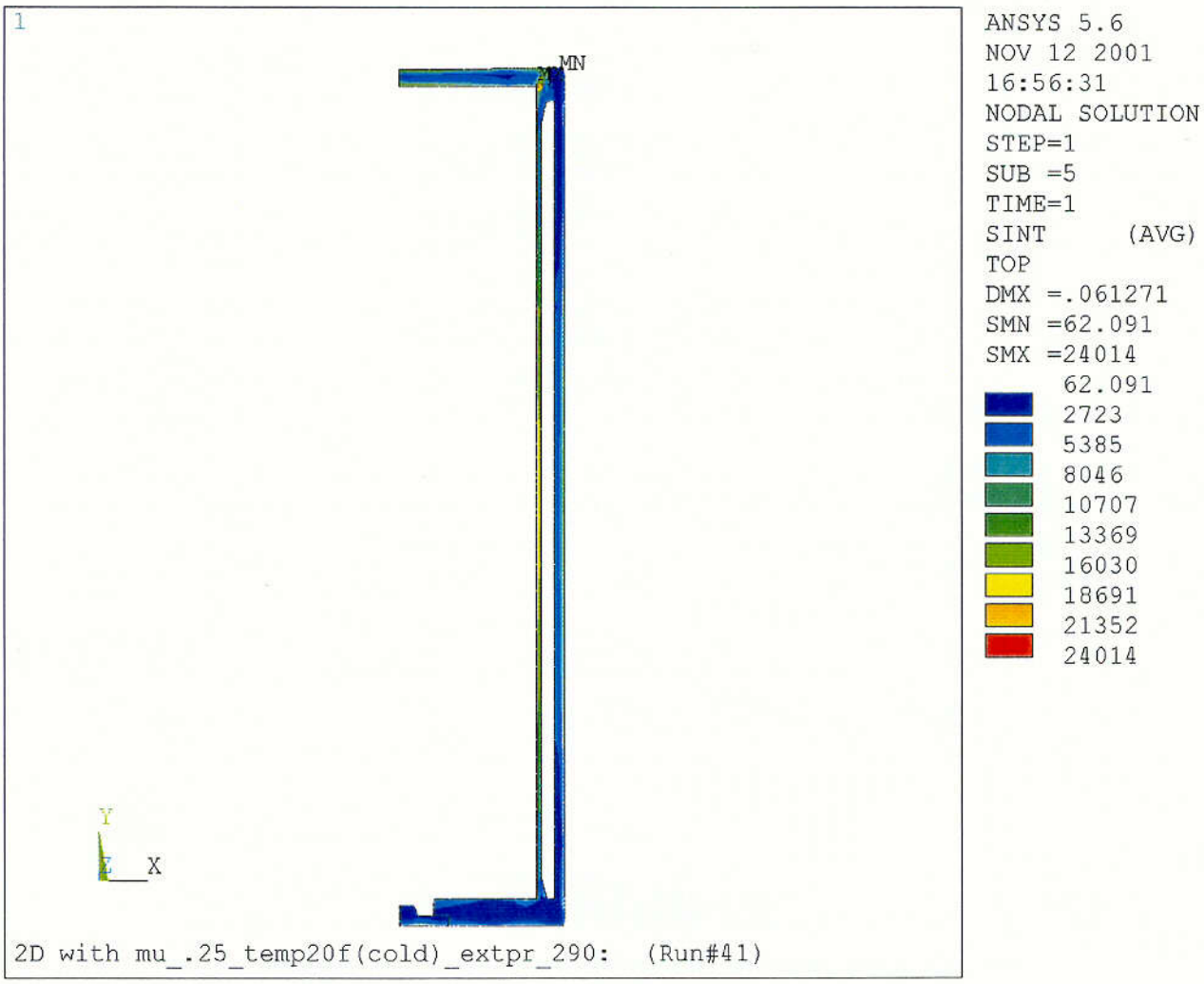
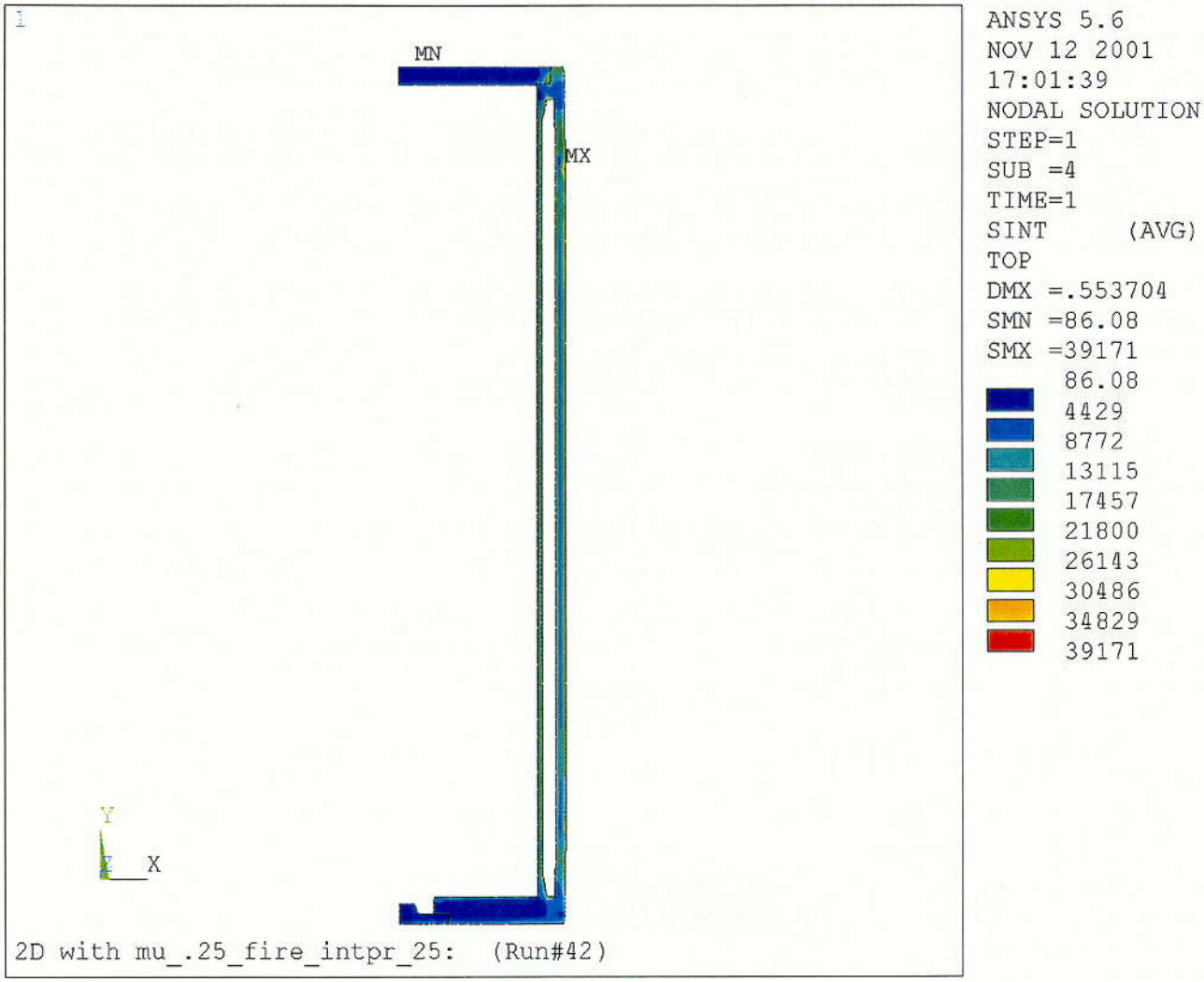



Figure 2.10.10-60
Run 42, Fire Accident, Cold Environment Stress Intensity Plot