## 1.0 OBJECTIVE

To demonstrate that the techniques employed in constructing the ANSYS finite element model of the 3-60B Cask represents the structure accurately and the element sizes used in the models converge to within a small tolerance of the theoretical values.

## 2.0 INTRODUCTION

The evaluations performed in the SAR of the 3-60B Cask (Reference 1 through 3) employ 3-dimensional finite element models for ANSYS software code (Reference 4). These models use 8-node solid elements (SOLID-185) for representing the cask components that do not undergo significant amount of bending deformation. The cask components that undergo significant amount of bending deformation have been represented by solid shell (SOLSH-190) elements in the models. Figure 1 shows a typical model of the 3-60B cask, identifying the element types employed in its modeling.

The modeling techniques employed in the 3-60B Cask model also utilize ANSYS contact elements to combine parts of the model with different grid densities. In order to model the bolting ring, which includes bolts holes, with octahedral elements, the bolting ring is modeled in two parts. These parts are combined together with the help of contact elements. Verification that this simplification does not compromise the stresses in the bolting ring is also included in this study.

The study is performed in two parts. In the first part, two finite element models of the bolting ring region- one identical to the SAR model and another model employing fine elements - are subjected to an internal pressure of 1,000 psi on the inner shell. The stresses in the two models are computed and compared with each other. It is shown that the stresses predicted by the SAR model are within 94% of the fine model.

In the second part of the study the effect of element size on stress prediction is studied. A simplified model of the inner shell is used for this study. The model represents a fixed-end inner shell that is subjected to 1,000 psi internal pressure. The number of elements through the thickness and along the longitudinal direction is varied and its effect on the end-moment and longitudinal stresses are plotted parametrically. The parameters corresponding to the inner shell are superimposed on these plots. It is shown that the element size corresponding to the SAR model can predict the end moment within 95% of the theoretical value. Models with these parameters will also predict the longitudinal stresses in the shell within 90% of a model with fine elements.

Additional comparison with the theoretical stress values has been performed with models representing the unit width of the inner shell element as a beam. The number of elements through the thickness and along the length is varied. The maximum longitudinal stresses are plotted parametrically. The parameters corresponding to the inner and outer shell model of the SAR have been superimposed. It is shown that these models are capable of predicting the stresses within 95% of the theoretical values.

## 3.0 REFERENCES

(1) Energy*Soutions* Document ST-501, Structural Analyses of the 3-60B Cask under Normal Conditions of Transport.

(2) Energy*Solutions* Document ST-502, Structural Analyses of the 3-60B Cask under Hypothetical Fire Accident Conditions.

(3) Energy*Solutions* Document ST-504, Structural Analyses of the 3-60B Cask under Drop Conditions.

(4) ANSYS, Rev. 11.0, Computer Software, ANSYS Inc., Canonsburg, PA, 2007.

(5) Formulas for Stress and Strain, Roark and Young, Fifth Edition, McGraw Hill Publications.

## 4.0 ANLYSIS DESCRIPTION

The studies performed in this document are divided into two parts. The first part includes a study to show that the simplifications employed in modeling techniques for representing the bolting ring-inner shell region in the 3-60B Cask ANSYS finite element model is appropriate. The second includes a study to show that finite element grid size employed in the inner-shell model of the 3-60B Cask accurately predicts the stresses.

The pertinent data of the finite element models used in this study is provided in Appendix 1 in printed form. The electronic data for all the analyses are provided in Appendix 2 on a DVD.

### 4.1 Modeling Technique Study

The 3-60B Cask finite element model representing the bolting ring-inner shell region has the following major characteristics.

- Uses ANSYS SOLIDSH-190 elements to represent the shells.
- Uses SOLID-185 elements to represent parts of the structure that do not undergo large bending deformation.
- Uses 1-element through the thickness to represent the shells.
- The shell elements at the interface are connected with single elements in the plate.
- The bolting ring is modeled in two parts that are tied together using bonded contact interface.

The objective of this study is to show that the model of the bolting ring-inner shell region constructed with the above characteristics appropriately represents the true nature of the region. To accomplish this two simplified finite element models of this region are constructed and analyzed for an internal pressure loading of 1,000 psi. The first model is similar to the model used in the 3-60B Cask SAR. This model is subsequently referred to as the "SAR Model" in the rest of the document. The second model is made with very fine elements and without the use of contact elements. This model is subsequently referred to as the "fine model" in the rest of the document. The simplified models have the following characteristics.

- They use an 11.25° (1/32nd symmetry) model.
- They use the bolting ring and a sufficient length (> $4\sqrt{Rt}$) of the inner shell in the model.
- They disregard the bolt-holes.
- They disregard the outer shell and the skirt part of the bolting ring.
- They are analyzed for 1,000 psi internal pressure loading on the inner shell.
- The maximum stress intensity, maximum axial stress, and linearized axial stress are computed for both the models.

The geometry of the bolting ring-inner shell region of the 3-60B Cask is shown in Figure 2. The region shown in solid colors in this figure is represented in the simplified finite element models.

4.1.1 <u>The SAR Model</u>

The SAR model finite element grid and the boundary conditions are shown in Figure 3. This model has the following additional characteristics (in addition to those described under the simplified model characteristics).

- It uses SOLSH-190 elements to represent the inner shell and keep the element size identical to the SAR model.
- It uses SOLID-185 elements to represent the plate part of the bolting ring, keeping the element size identical to the SAR model everywhere except in the bolt-hole area.
- It models the bolting ring in two parts and joins the two parts by bonded contact interface, identical to the SAR model.

The stress intensity contour plot of the SAR model is shown in Figure 4. The stress intensity contour plot in the inner shell is shown in Figure 5 and the contour of the longitudinal stresses in the inner shell is shown in Figure 6. The linearized membrane and membrane plus bending plots over the inner shell cross-section are shown in Figure 7.

4.1.2 <u>The Fine Model</u>

The fine model finite element grid geometry is shown in Figure 8. This model has the following additional characteristics (in addition to those described under the simplified model characteristics).

- It uses SOLID-185 elements to represent the entire model.
- It uses 5-elements through the thickness of the shell.
- It uses very fine elements to represent the plate near the interface region.
- It models the plate in one piece. No bonded contact is used.

The stress intensity contour plot of the fine model is shown in Figure 9. The stress intensity contour plot in the inner shell is shown in Figure 10 and the contour of the longitudinal stresses in the inner shell is shown in Figure 11. The linearized membrane

and membrane plus bending plots over the inner shell cross-section are shown in Figure 12.

4.1.2 Results and Comparison

The results of the SAR model and the fine model analyses are presented in Table 1. The comparison of the results and percentage difference of the results between the two models are also shown in this table. The following conclusions are made from the comparison of the results of the two models.

- The SAR Model computes the maximum stress intensity within 94% of the Fine Model.
- The SAR Model computes higher axial stresses than the Fine Model. It is +9% for the maximum value and +5% for the linearized membrane + bending value.
- Inclusion of bonded contact interface at the location chosen by EnergySolutions in the SAR model does not have any significant effect. The stress values and pattern are very similar to each other (see Figures 4 & 9). Also the interface is located at a relatively lightly loaded location of the bolting ring.

## 4.2 Grid Sensitivity Study

In order to demonstrate that the finite element grid used in the ANSYS model of the 3-60B Cask adequately represents the bending behavior of the shell, a grid sensitivity study is performed. For this purpose a sufficiently large length ( $>4\sqrt{Rt}$ ) of the inner shell, fixed at one of its edges and free on the other, is analyzed for an internal pressure loading of 1,000 psi. The models are constructed from solid shell (SOLSH-190) elements - the same elements used in the 3-60B Cask ANSYS model to represent the inner shell. Each model has a length of 25" and extends $10^{o}$ in the circumferential direction. Figure 13 shows the geometry of the model.

Various grid densities are obtained by varying the number of elements through the thickness and through the length. Permutations of 1, 2, 3, and 4 elements through the thickness and 10, 15, 20, 25, 30, and 40 elements through the length is used to encompass grid densities that range between "coarse" and "fine". All the 24 (4×6) models resulting from the aforementioned permutations are analyzed for end moments and maximum longitudinal stresses. The longitudinal stress data is linearized over the shell thickness at the fixed-edge for each model. An input file written in ANSYS programming language (APL) is used to automate the computation process. This file is included in Appendix 2 of this document.

The contour plot of the longitudinal stress for a typical model (4 elements through the thickness; 40 elements through the length) is shown in Figure 14. The end-moments from each model analysis are plotted in Figure 15 in a parametric form. This plot also shows the length-to-thickness ratio of the finite element size used in the 3-60B Cask ANSYS model. It is observed that the end-moment is not sensitive to the number of elements through the thickness or the length direction. The plot also shows a "theoretical value"

calculated from the formulas obtained from Reference 5. It should be noted that this "theoretical value" is not precise and is given here for reference purpose only.

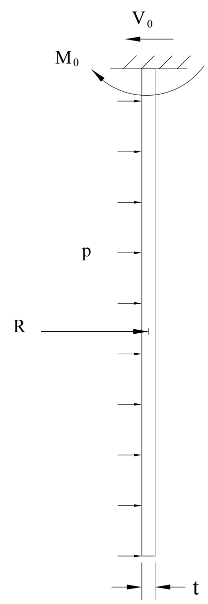
Using the formulas from Reference 5 the theoretical end moment is obtained as follows:

R = Mean radius of the shell = 17.875 in

t = Shell thickness = 0.75 in

E = Young's modulus of the material = $30\times10^6$ psi

ν = Poisson's ratio of the material = 0.3




$$D = \frac{Et^3}{12(1-\nu^2)} = \frac{30\times10^6\times0.75^3}{12\times(1-0.3^2)} = 1.159\times10^6 \text{ in-lb}$$

$$\lambda = \left[\frac{3\times(1-\nu^2)}{R^2t^2}\right]^{0.25} = \left[\frac{3\times(1-0.3^2)}{17.875^2\times0.75^2}\right]^{0.25} = 0.3511\ /\text{in}$$

Radial displacement due to p, $V_0$, and $M_0$ are:

$$\Delta R_p = \frac{pR^2}{Et}$$

$$\Delta R_{V_0} = \frac{-V_0}{2D\lambda^3}$$

$$\Delta R_{M_0} = \frac{M_0}{2D\lambda^2}$$

Rotation due to $V_0$ and $M_0$ are:

$$\psi_{V_0} = \frac{V_0}{2D\lambda^2}$$

$$\psi_{M_0} = \frac{-M_0}{D\lambda}$$

For the fixed boundary conditions:

$$\Delta R_p + \Delta R_{V_0} + \Delta R_{M_0} = 0 \quad \text{.......................................}(1)$$

$$\psi_{V_0} + \psi_{M_0} = 0 \quad \text{.......................................}(2)$$

Solving equations (1) and (2) simultaneously, one gets:

$$M_0 = \frac{p}{2\lambda^2} = \frac{1{,}000}{2\times0.3511^2} = 4{,}056 \text{ in-lb/in}$$

The longitudinal membrane + bending stresses, linearized over the thickness, obtained from various models are plotted in Figure 16 in a parametric form. Taking the value corresponding to the fine model (4 elements through the thickness; 40 elements through the length) as the datum, 90 and 95 percentile lines are shown in this figure. It is observed that the membrane + bending stress corresponding to the size of elements used in the 3-

60B Cask ANSYS model yields a result that is within approximately 90% of the fine model result.

4.2.1 Comparison with the Theoretical Stresses

In order to compare the stresses calculated by the finite element models of various grid densities that use solid shell (SOLSH-190) elements with the theoretical value, cantilever beam model, as shown in Figure 17, was analyzed for various grid densities. The choice of this model was influenced by the fact that the theoretical stress for this case is precise and is not strongly dependent on the thickness of the cross-section. A 1"×1" cross-section and 10" length of the beam was used for modeling. The top surface of the beam was subjected to a uniform pressure of 100 psi. Under this loading the bending moment at the support is:

$$M = 1\times10\times100\times10/2 = 5{,}000 \text{ in-lb}$$

The maximum bending stress is:

$$\sigma = 6\times5{,}000/1 = 30{,}000 \text{ psi}$$

The finite element model is constructed using SOLSH-190 elements and the following material properties.

$$E = 30\times10^6 \text{ psi, and } \nu = 0.3$$

Various grid densities are obtained by varying the number of elements through the thickness and through the length. Permutations of 1, 2, 3, 4, 6 and 8 elements through the thickness and 5, 10, 15, 20, 30, and 40 elements through the length is used to encompass grid densities that range between "coarse" and "fine". All the 36 (6×6) models resulting from the aforementioned permutations are analyzed. An input file written in ANSYS programming language (APL) is used to automate the computation process. This file is included in Appendix 2 of this document.

Contour plot of longitudinal stresses in a typical model is shown in Figure 18. The maximum longitudinal stresses obtained from each model analysis are plotted in Figure 19 in a parametric form. To prevent over-crowding this plot, results from 6 and 8 elements through the thickness are not included in this plot since they are very close to the corresponding results from the 4 element through the thickness solution.

Figure 20 shows a plot of 1-element through the thickness plot for various element sizes. The theoretical result is also superimposed in this plot along with the 95 and 90 percentile of the theoretical result band. The element size of the shell elements at the bolting ring-shell interface region is 0.9375" (from Reference 1 through 3). The inner shell is 0.75" thick and the outer shell is 1.25" thick. Thus the length-to-thickness ratios of the two shells are:

Inner shell element length = 0.9375/0.75 = 1.25t

Outer shell element length = 0.9375/1.25 = 0.75t

Figure 20 also indicates these element length-to-thickness ratios. It is seen that for the pure bending problem the element size chosen for the inner and outer shells near the bolting ring interface region is expected to produce results within 95% of the theoretical value.

As an additional check a finite element model of a unit width of the inner shell, having the same element sizes as used in the 3-60B Cask ANSYS model is analyzed as a cantilever beam. The beam has the following geometric properties.

Width = 1", Thickness = 0.75", Length = 10"

It is subjected to a uniform pressure of 56.25 psi over the top surface. Under this loading,

$$M = 1\times10\times56.25\times10/2 = 2{,}812.5 \text{ in-lb}$$

The maximum bending stress is:

$$\sigma = 6\times2{,}812.5/0.75^2 = 30{,}000 \text{ psi}$$

The finite element model is constructed using SOLSH-190 elements and the following material properties.

$$E = 30\times10^6 \text{ psi, and } \nu = 0.3$$

The longitudinal stress contour plot of this model is shown in Figure 21. The maximum stress obtained from this model is 29,971 psi, which is within 99.9% of the theoretical value.

## 5.0 CONCLUSIONS

From the analyses provided in this document it has been demonstrated that the modeling techniques employed in constructing the ANSYS finite element models, used for the analyses of the 3-60B Cask, adequately represent the cask structure in the most vulnerable area (the bolting ring- inner/outer shell region). It has been further demonstrated that the selection of the element sizes in this region will predict the results that are within small tolerances of the corresponding theoretical values.

Table 1 - Comparison of Results of the SAR & the Fine Models

| Comparison of Results | | | |
|---|---|---|---|
| Quantity | SAR Model | Fine Model | % Difference(1) |
| Maximum Deflection (in) | 0.0149(2) | 0.014682(3) | +1.48% |
| Maximum S.I. in the joint (psi) | 28,093(2) | 29,741(3) | -5.54% |
| Maximum S.I. in the shell (psi) | 28,093(4) | 29,741(5) | -5.54% |
| Maximum Sigz in the shell (psi) | 21,118(6) | 19,356(7) | +9.10% |
| Linearized Mem+Bend Sigz (psi) | 21,118(8) | 20,124(9) | +4.94% |

NOTES:

(1) Percentage difference is calculated with respect to the fine model value.

(2) See Figure 4.

(2) See Figure 9.

(3) See Figure 5.

(4) See Figure 10.

(5) See Figure 6.

(6) See Figure 11.

(7) See Figure 7.

(8) See Figure 12.

**Figures**

(21 Pages)

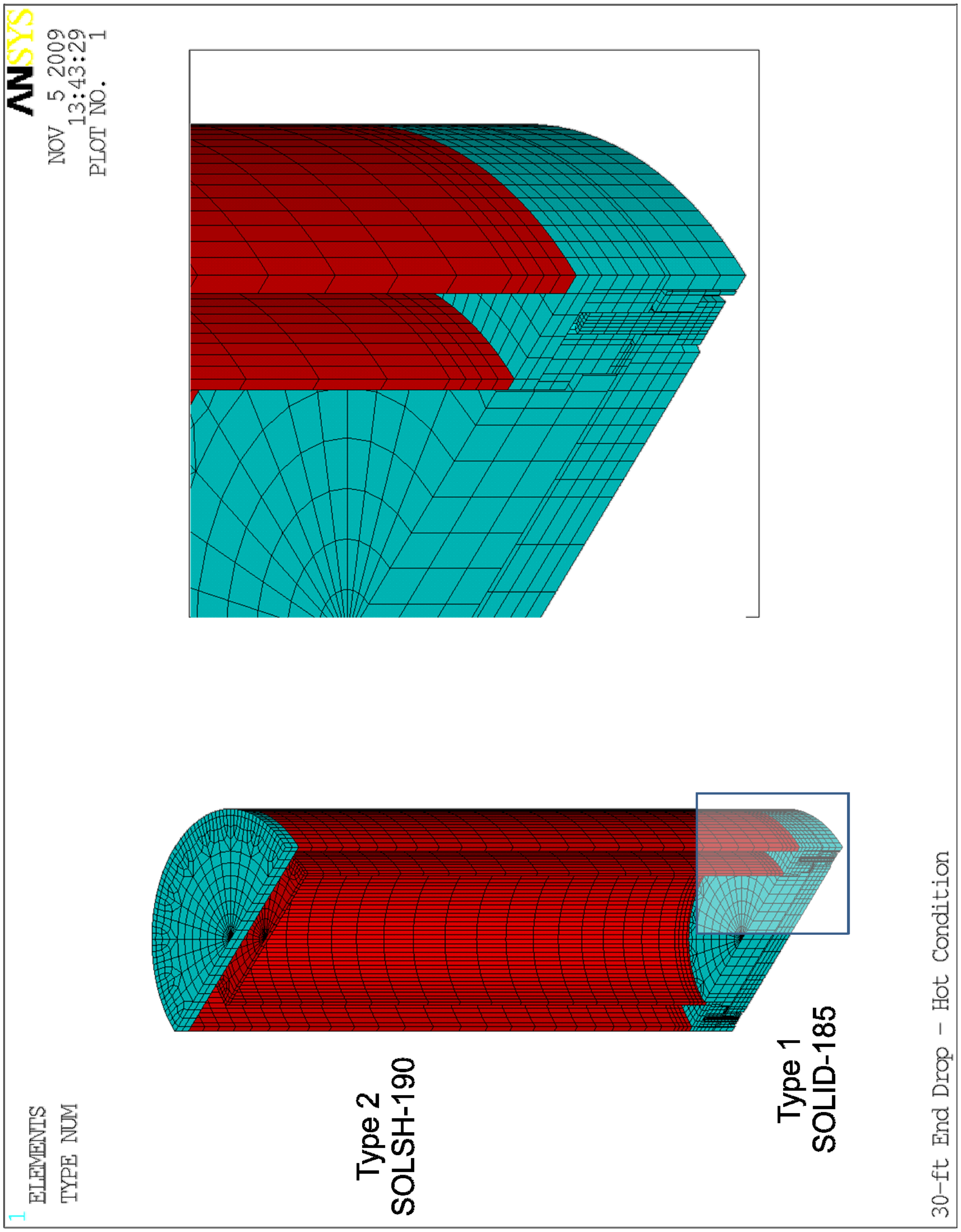



Figure-1

3-60B Cask FEM - Element Types Used to Model Structural Components

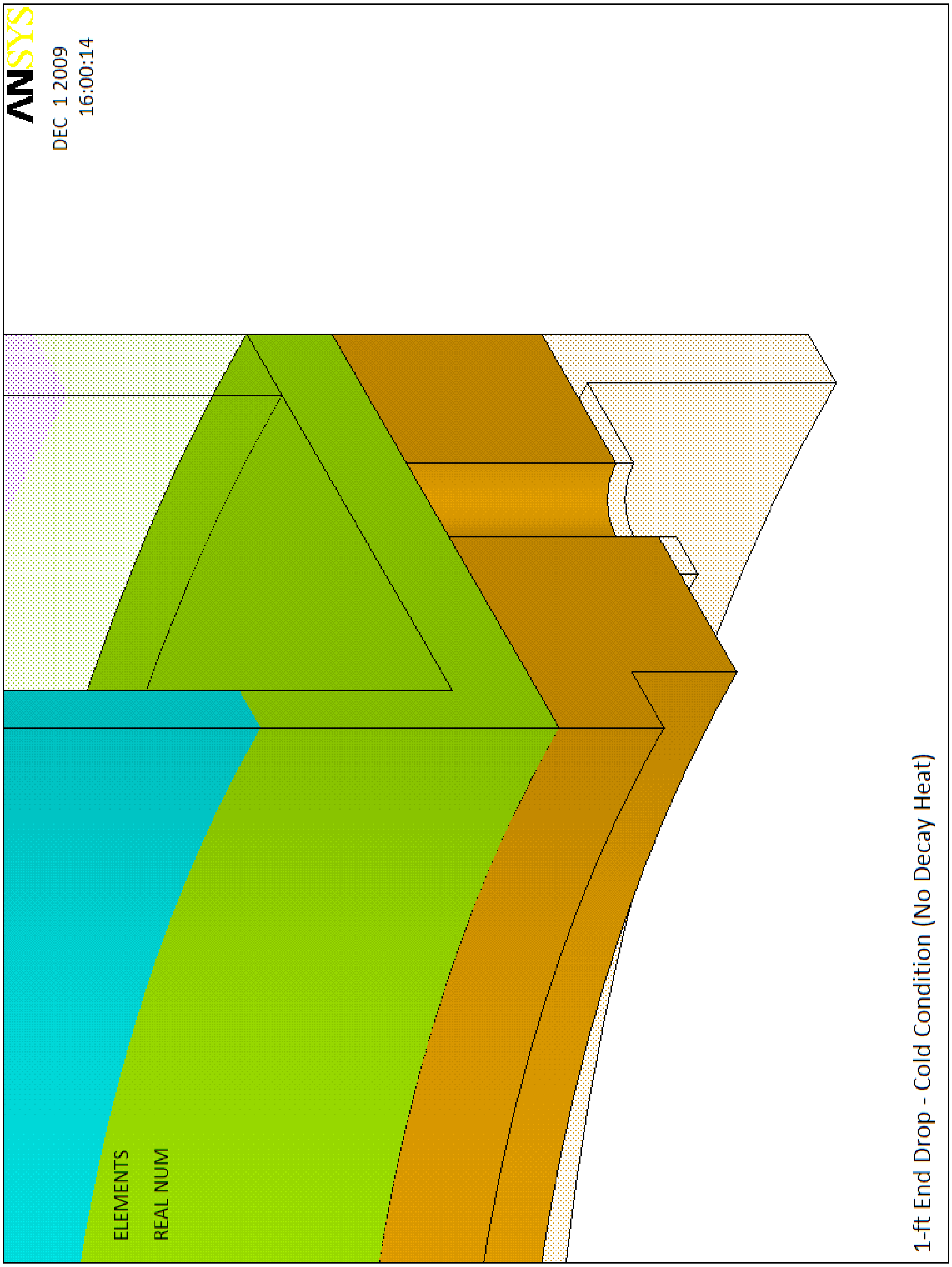



Figure-2
3-60B Cask FEM - Bolting Ring Area Modeling Details

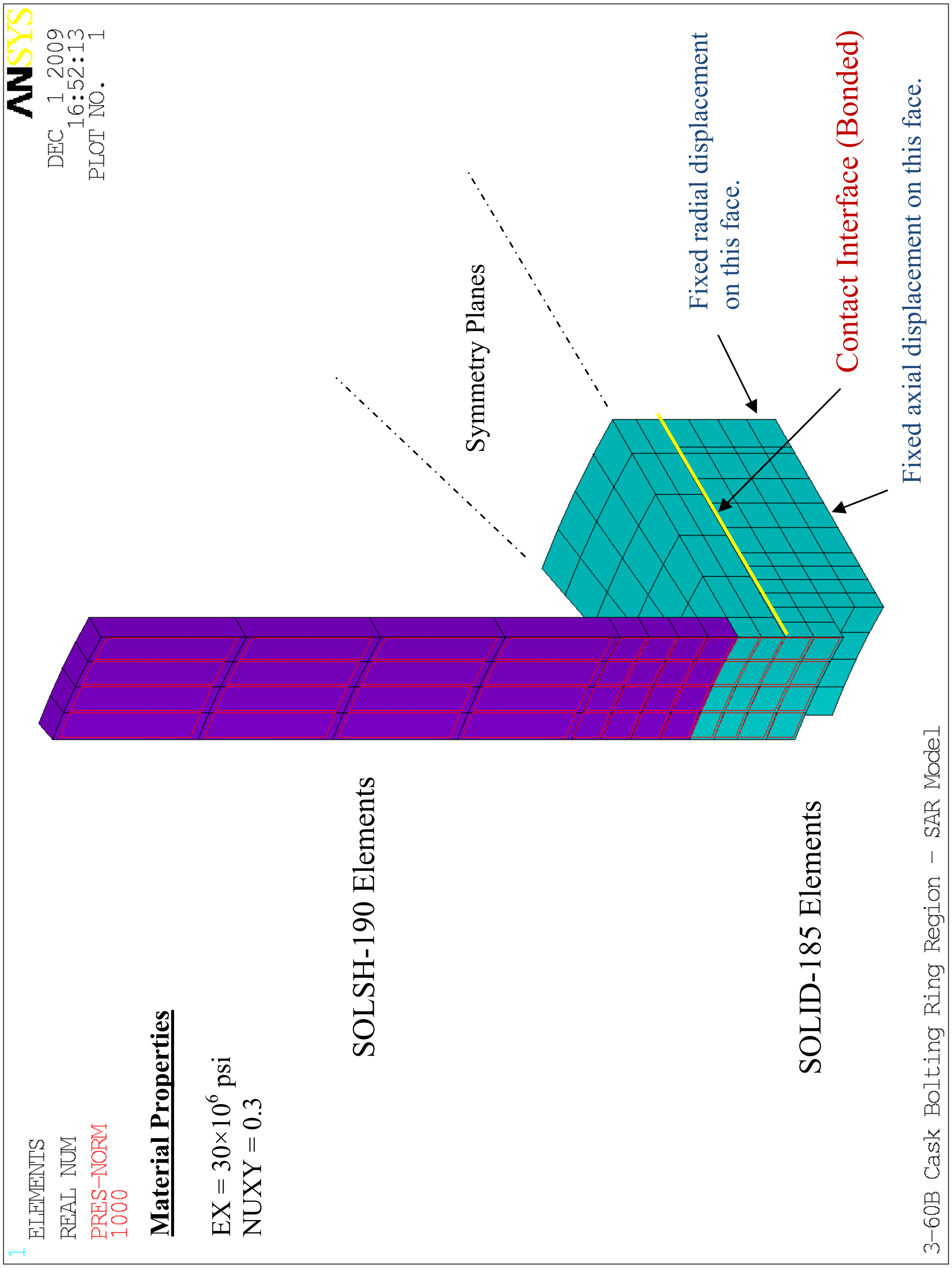



Figure-3

Simplified Bolting Ring Area - SAR Model

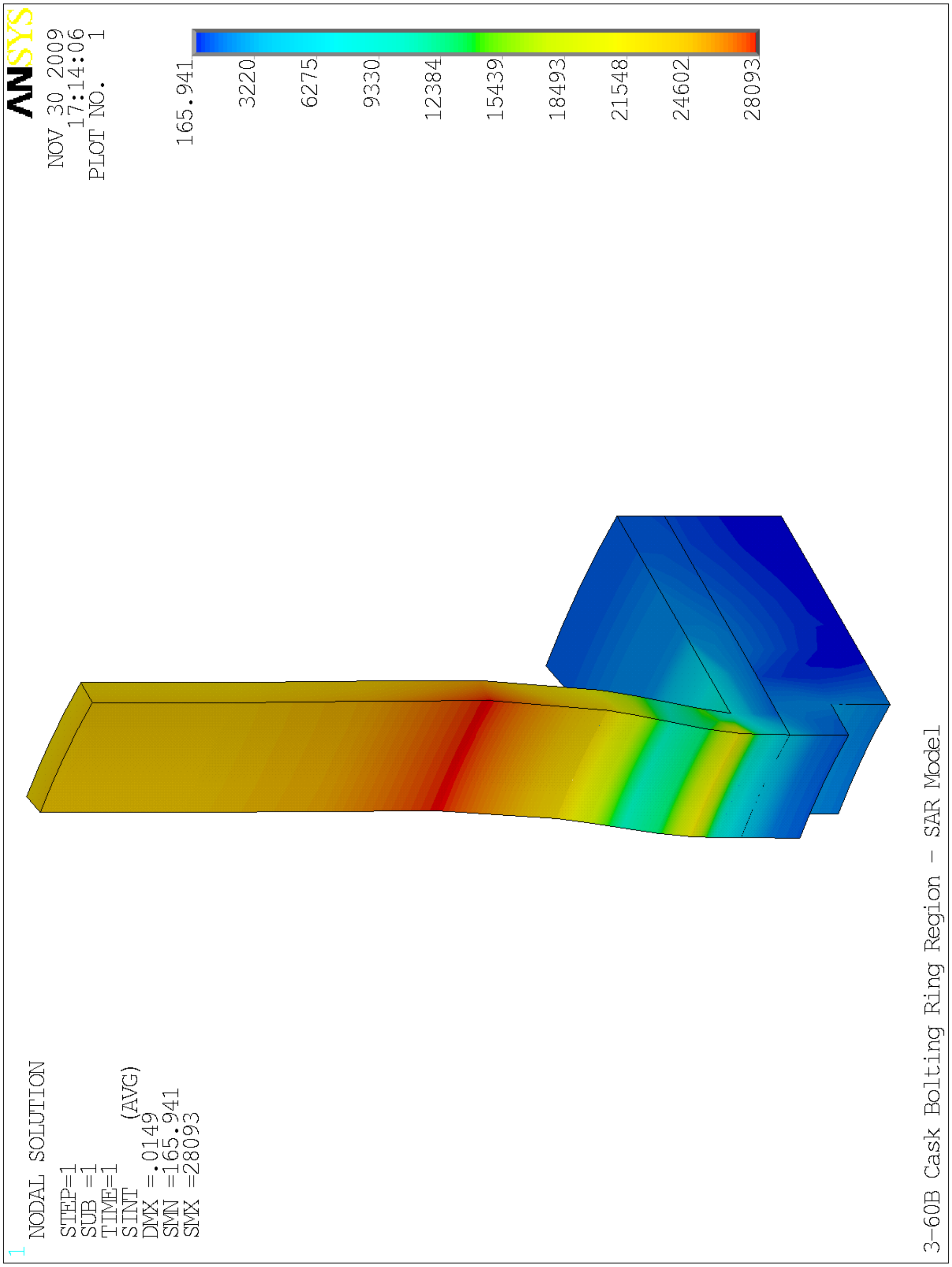



Figure-4

SAR Model - Stress Intensity Contour Plot

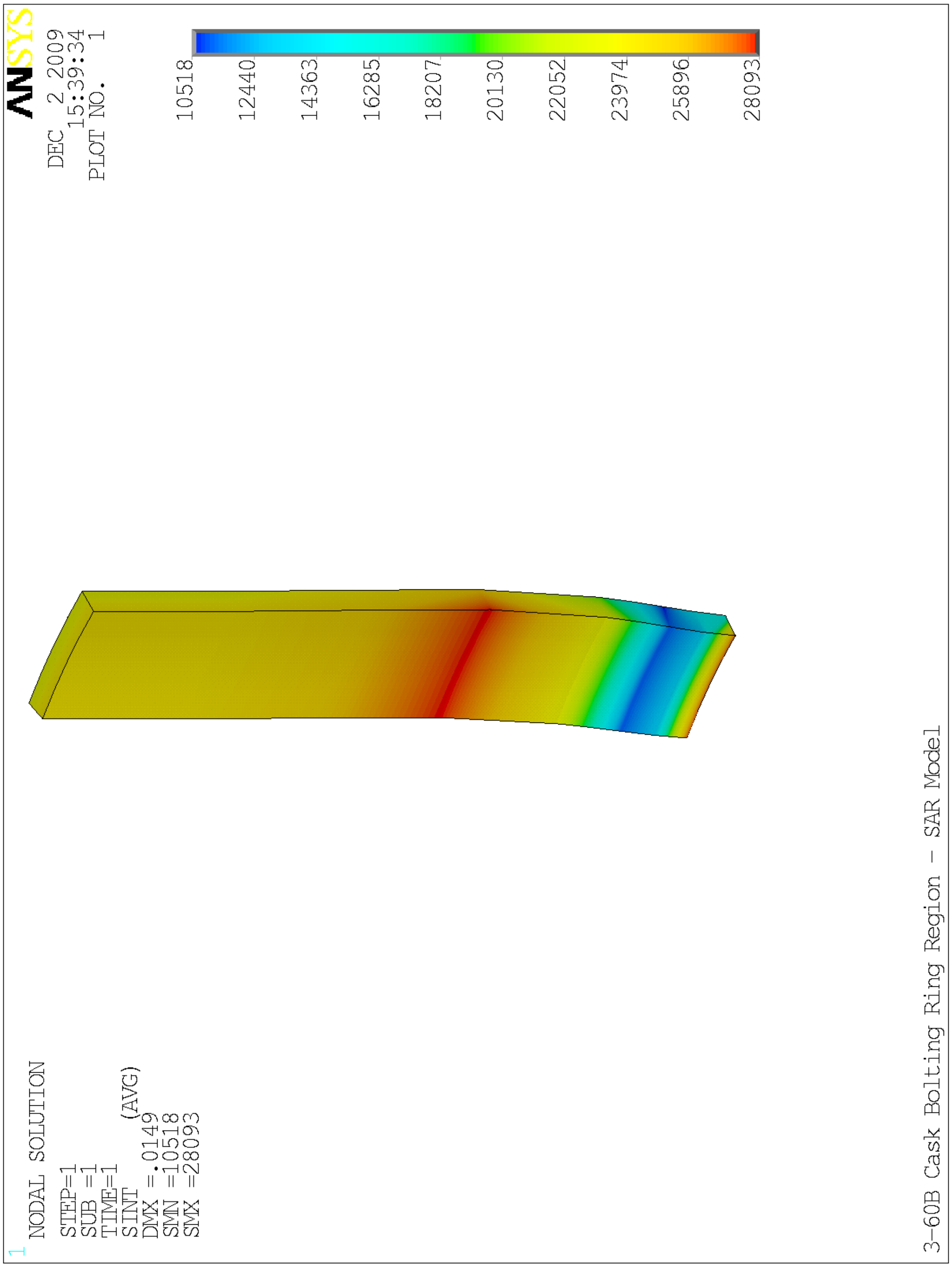



Figure-5

SAR Model - Shell Stress Intensity Contour Plot

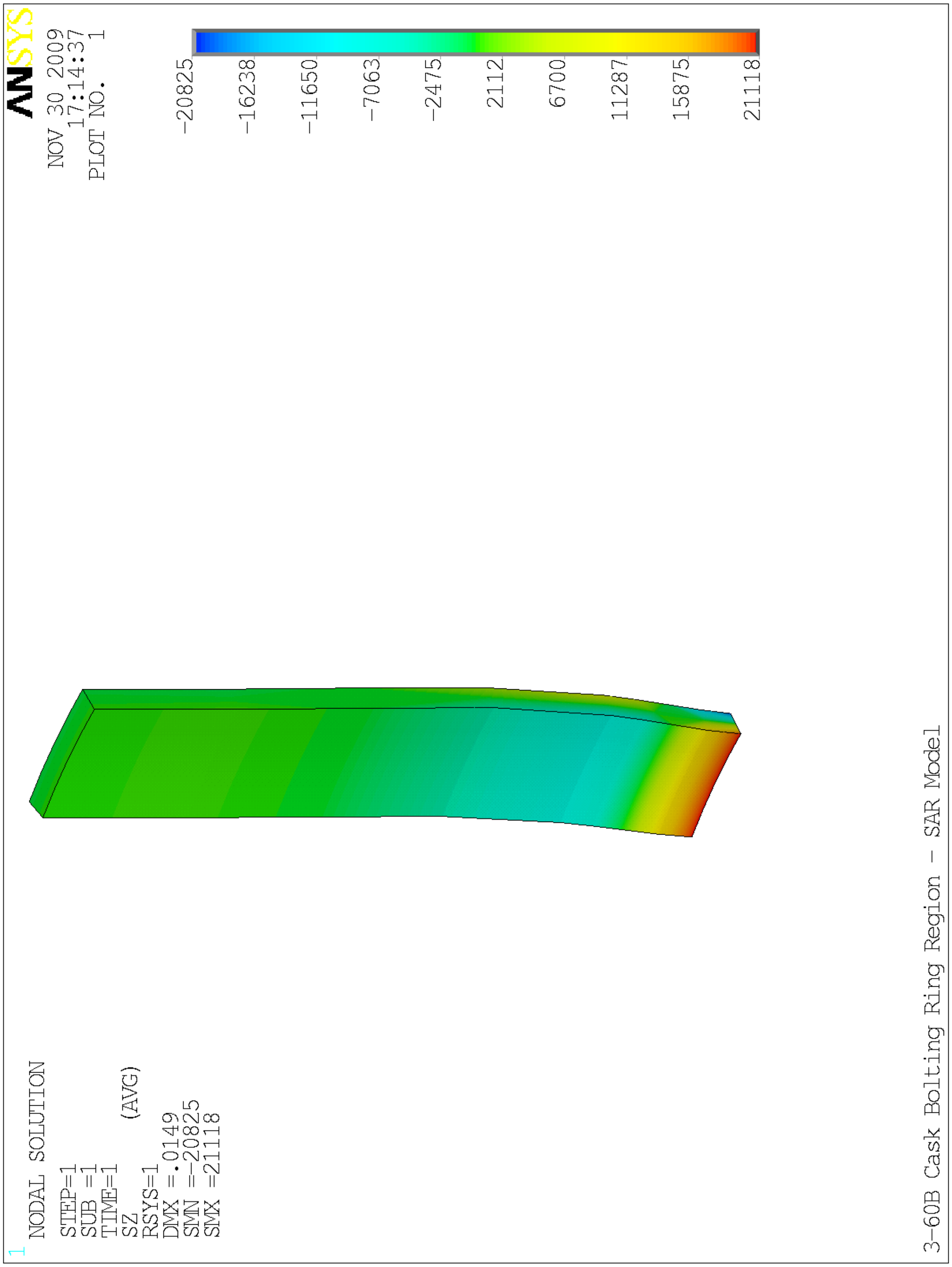



Figure-6

SAR Model - Shell Longitudinal Stress Contour Plot

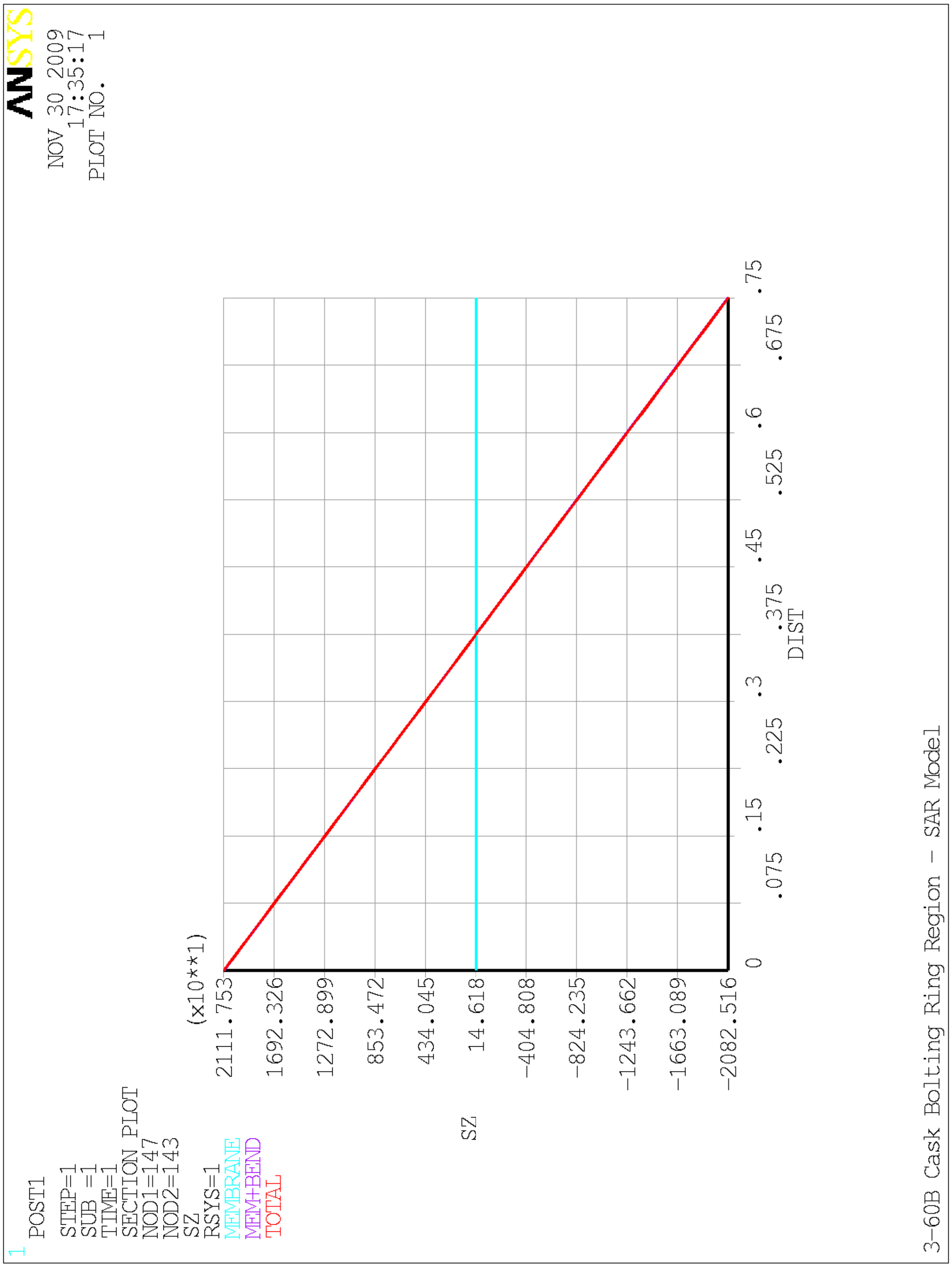



Figure-7

SAR Model - Linearized Longitudinal Stress in the Shell

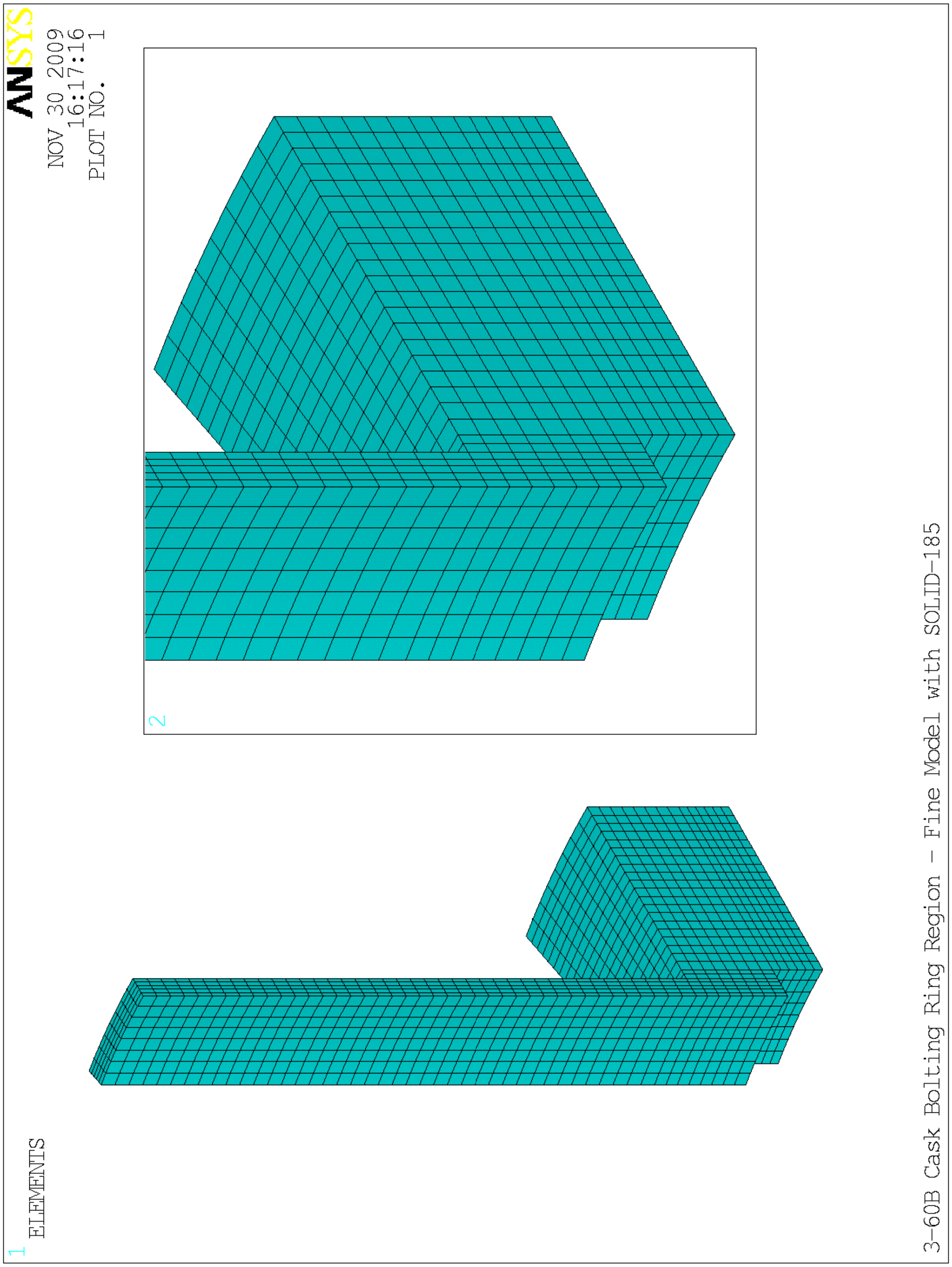



Figure-8

Simplified Bolting Ring Area - Fine Model

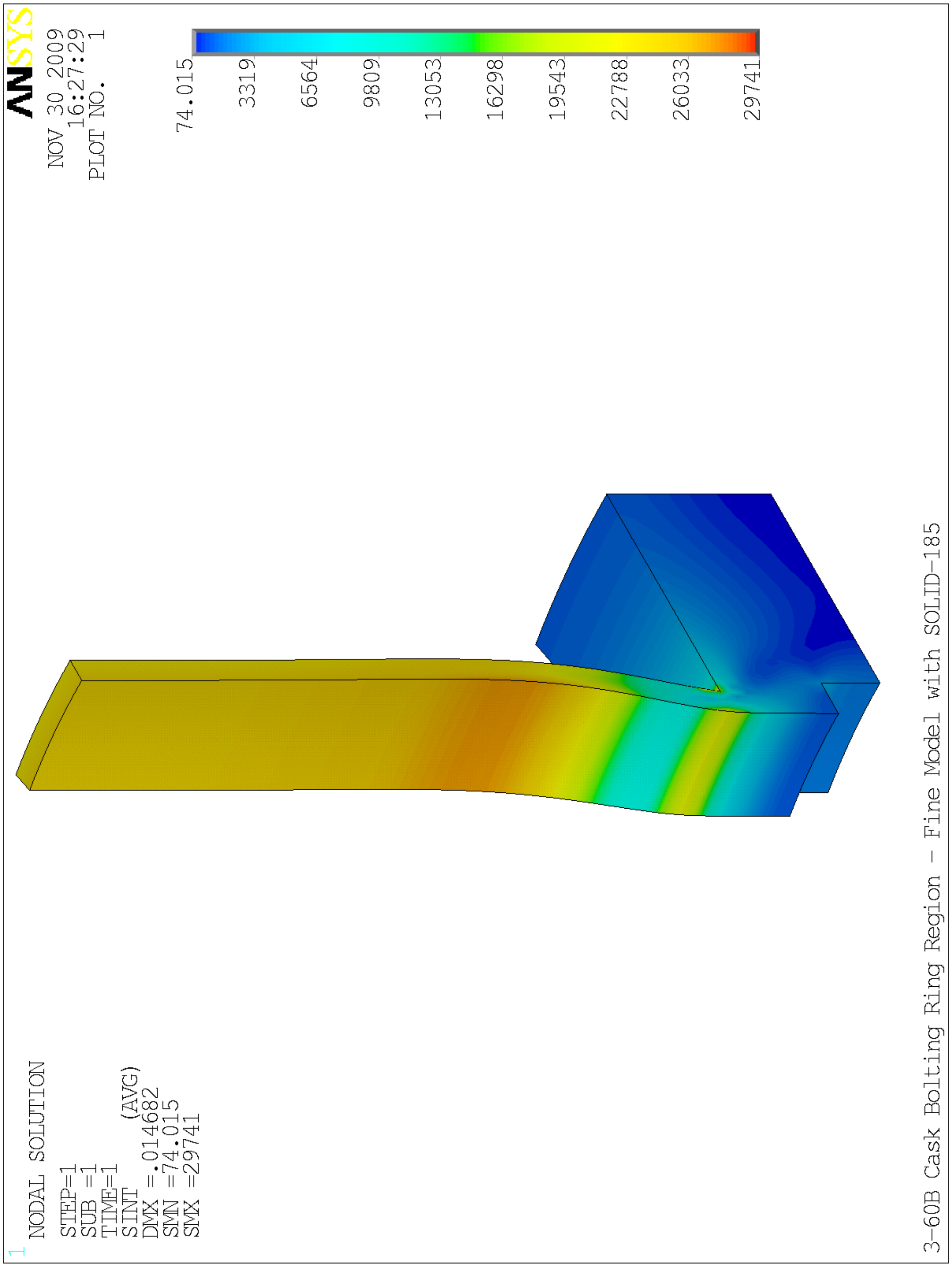



Figure-9

Fine Model - Stress Intensity Contour Plot

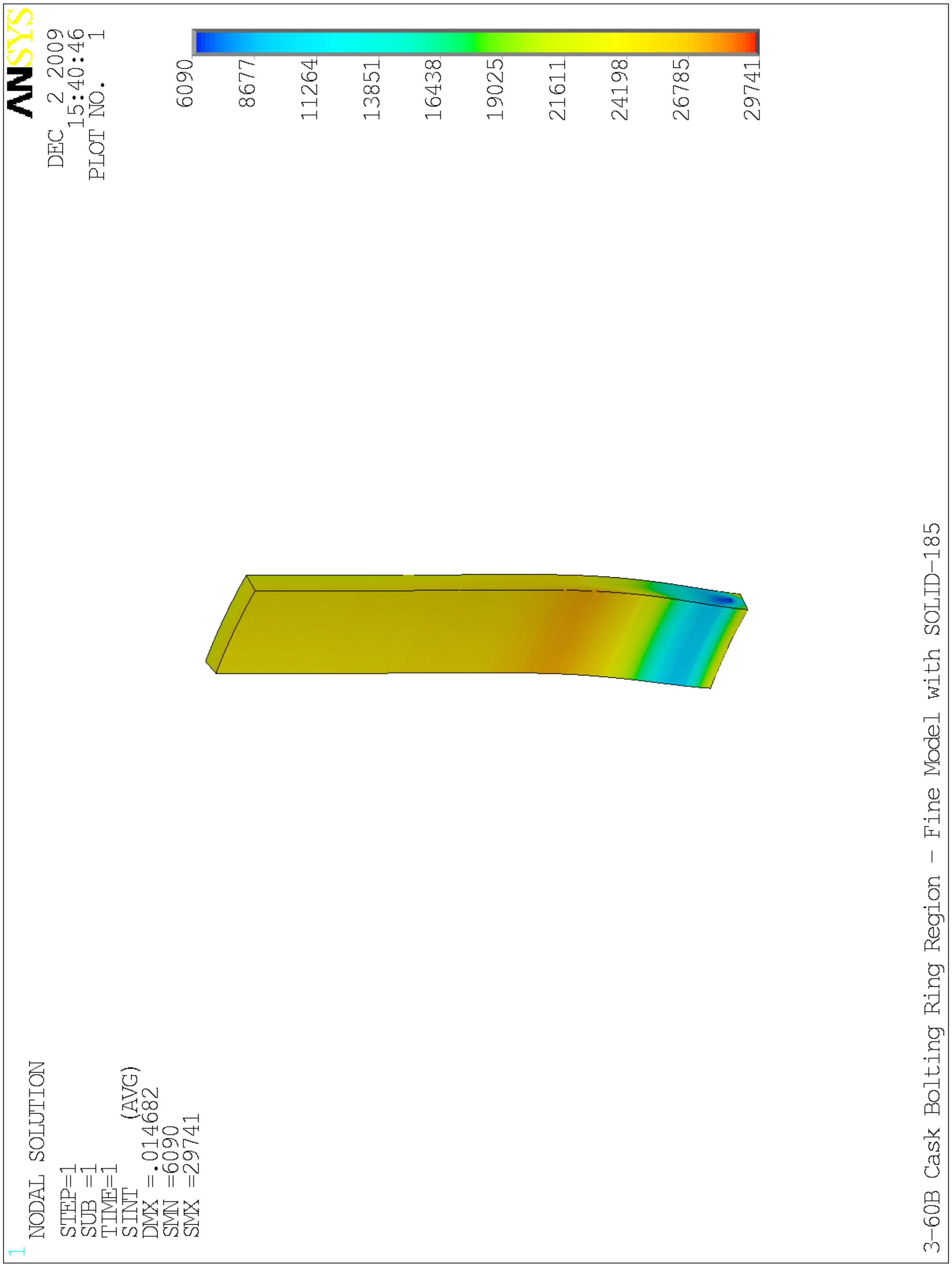



Figure-10

Fine Model - Shell Stress Intensity Contour Plot

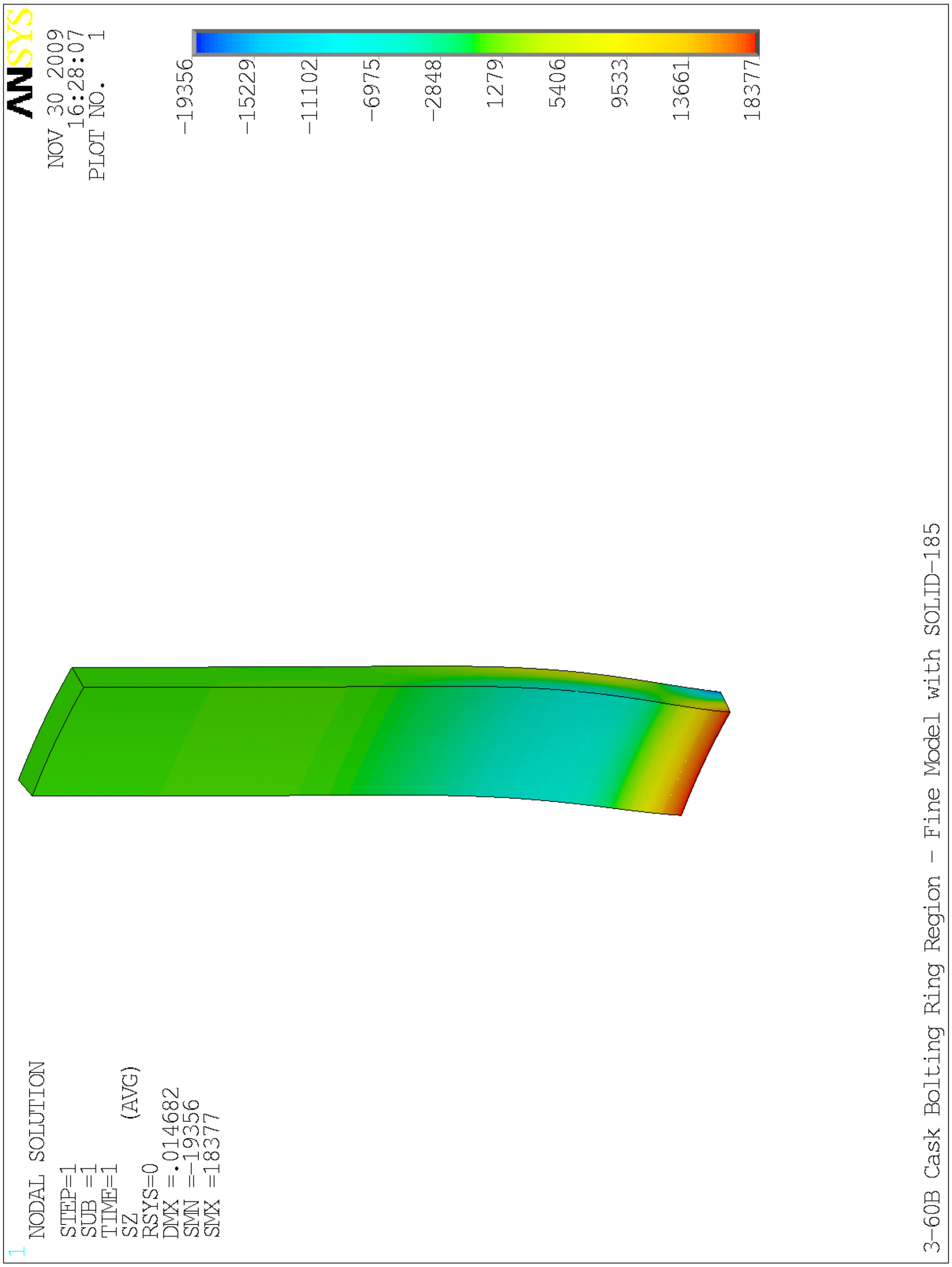



Figure-11

Fine Model - Shell Longitudinal Stress Contour Plot

| | |
|---|---|
| **Title** 3-60B Cask ANSYS Finite Element Model Grid Convergence Study | |
| **Calc. No.** ST-608 **Rev.** 0 | **Sheet** 12 of 21 |

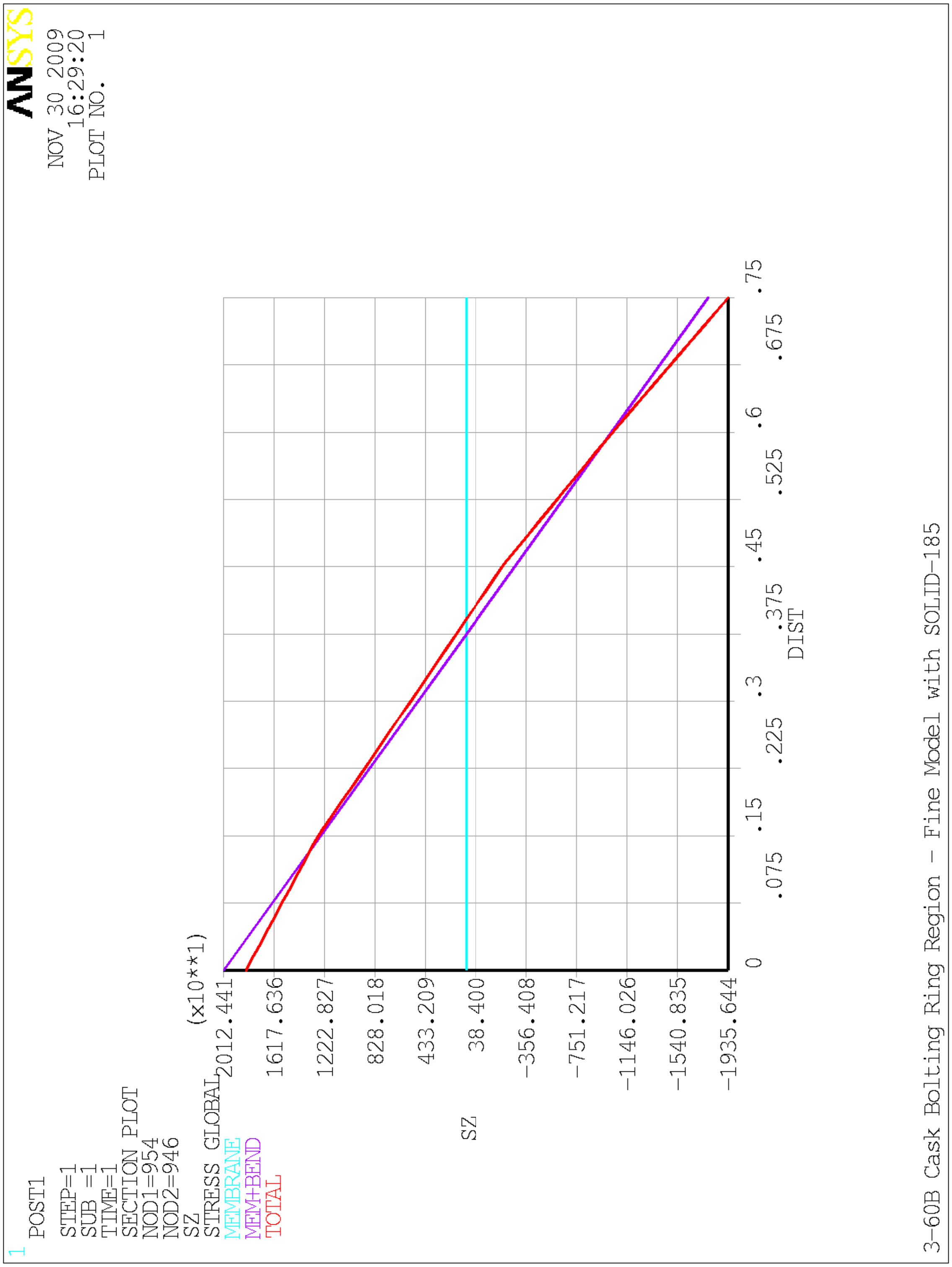



Figure-12

Fine Model - Linearized Longitudinal Stress in the Shell

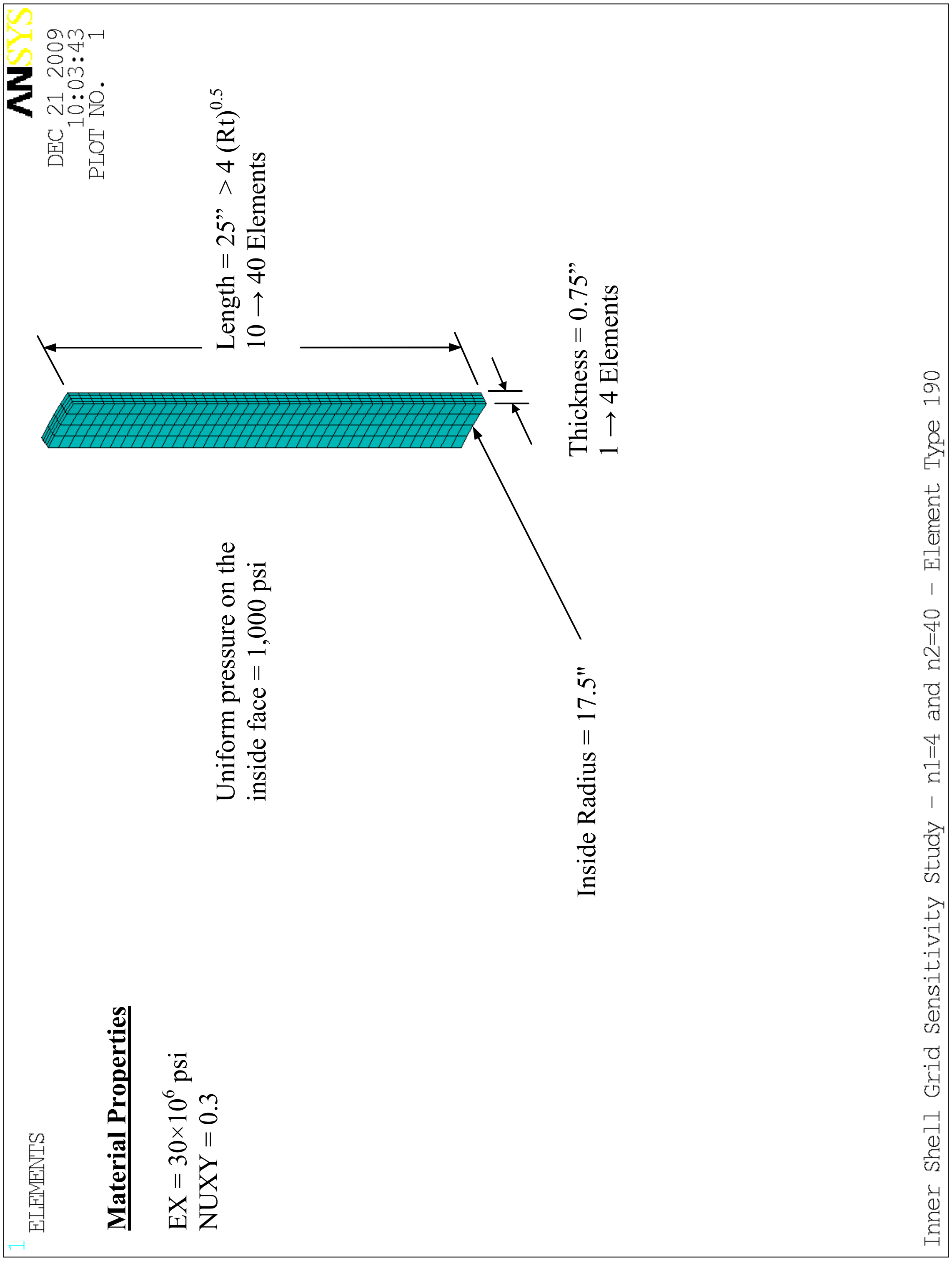



Figure-13

Shell Modeling Grid Sensitivity Study - FEM Dimensions

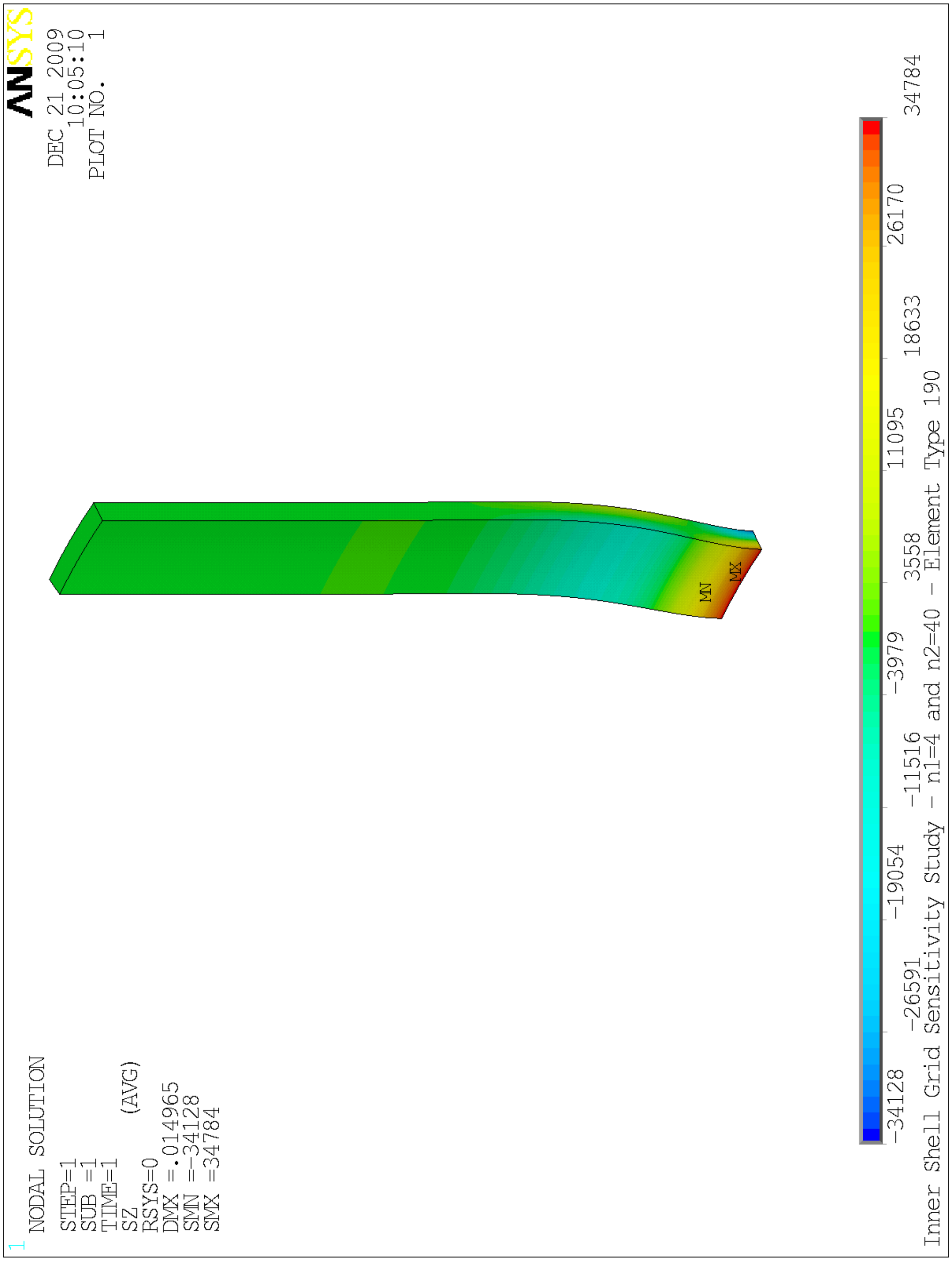



Figure-14

Grid Sensitivity Study - A Typical Longitudinal Stress Contour Plot

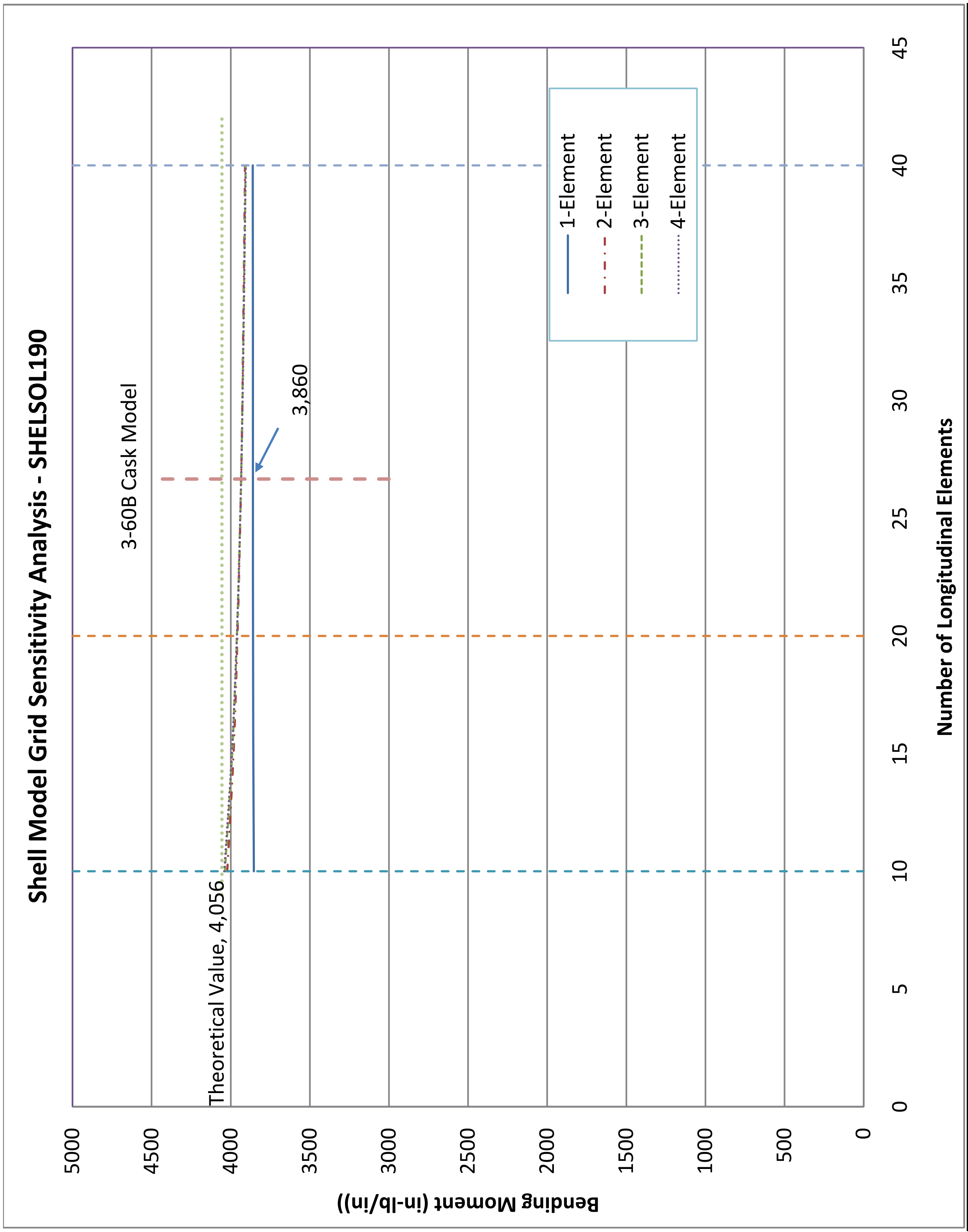



Figure-15

Grid Sensitivity Study - End Moment Plot for Models with Various Grid Densities

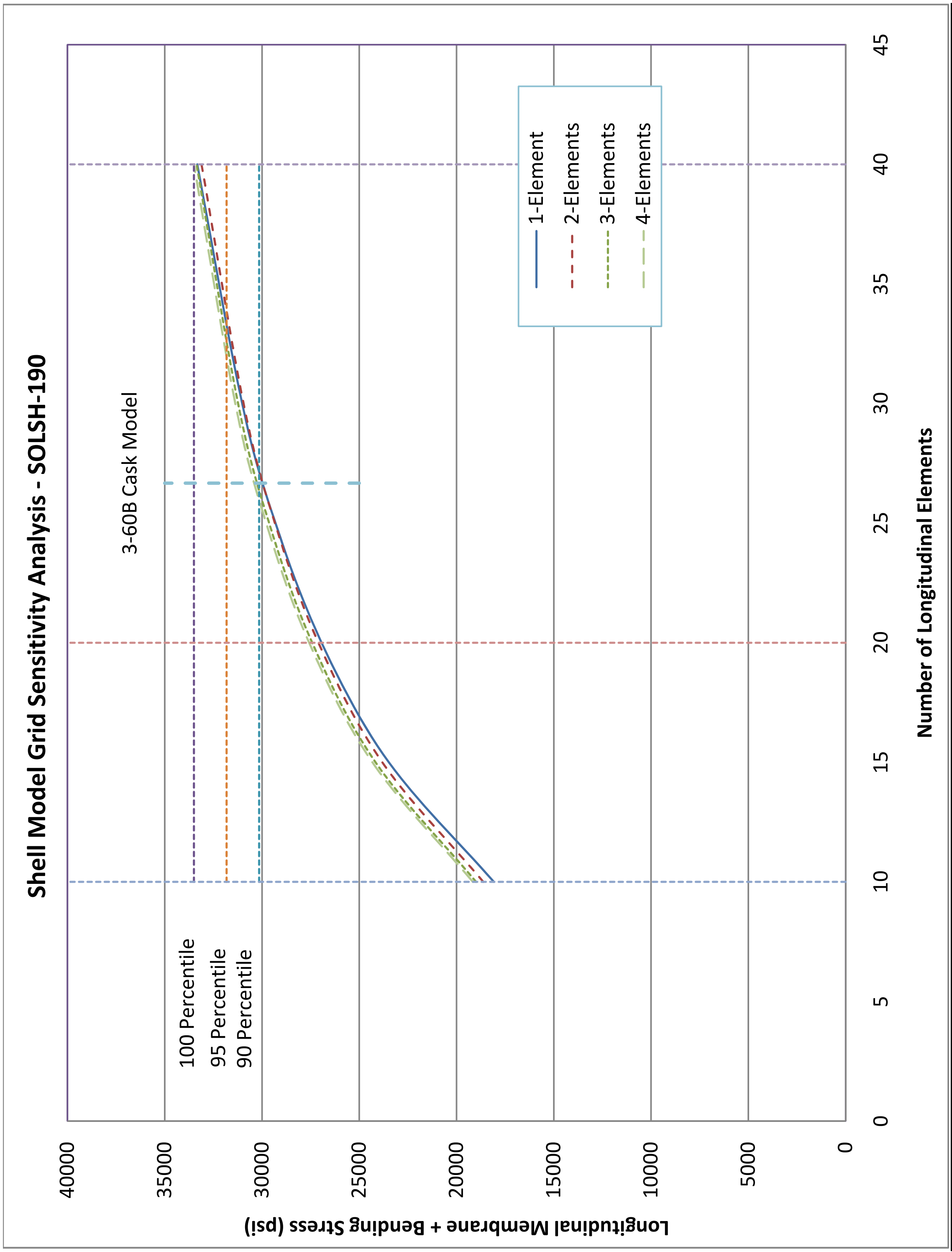



Figure-16

Grid Sensitivity Study - Linearized Longitudinal Bending Stress Plot for Models with Various Grid Densities

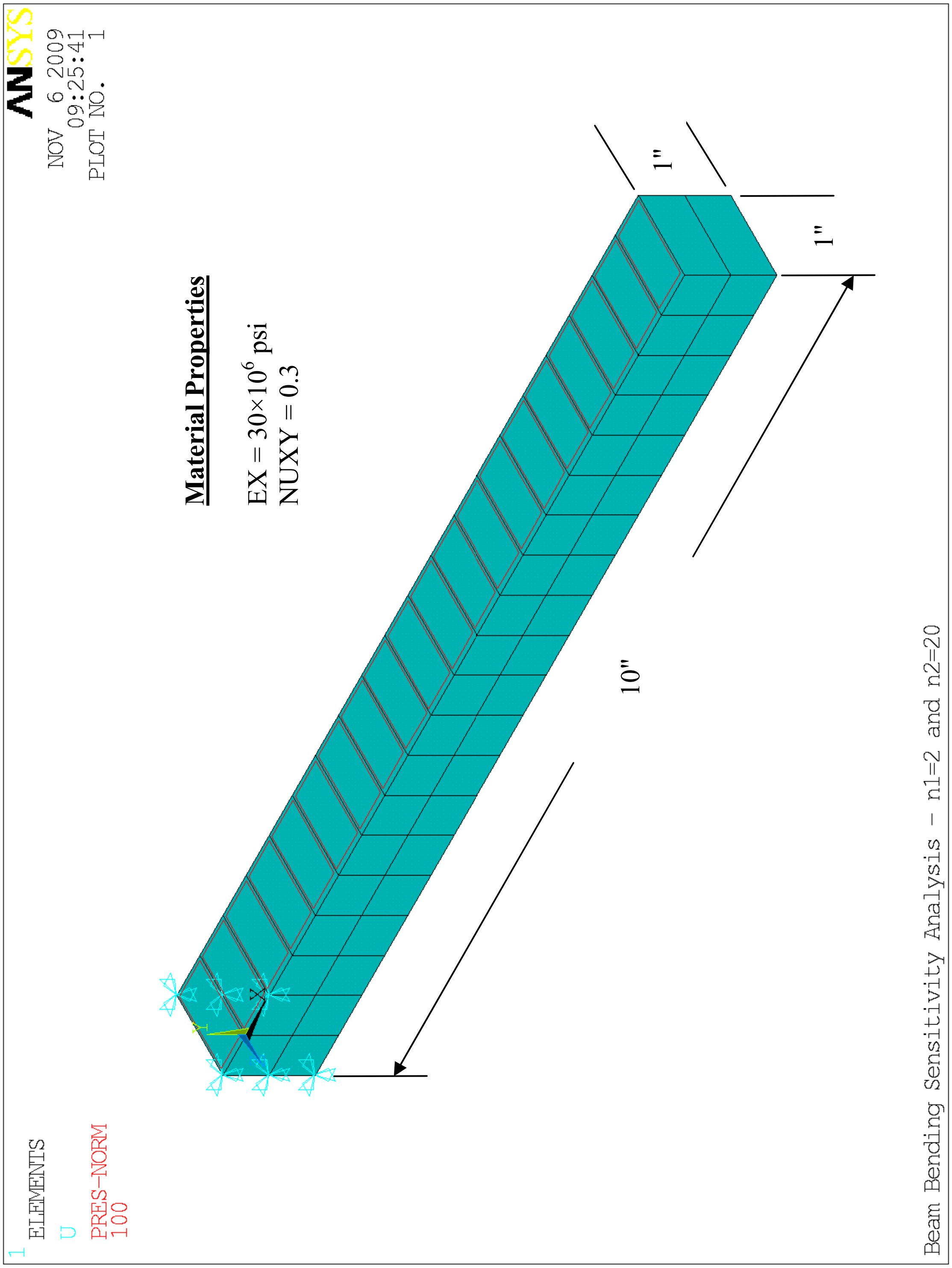



Figure-17

Beam Model Grid Sensitivity Study - FEM Dimensions

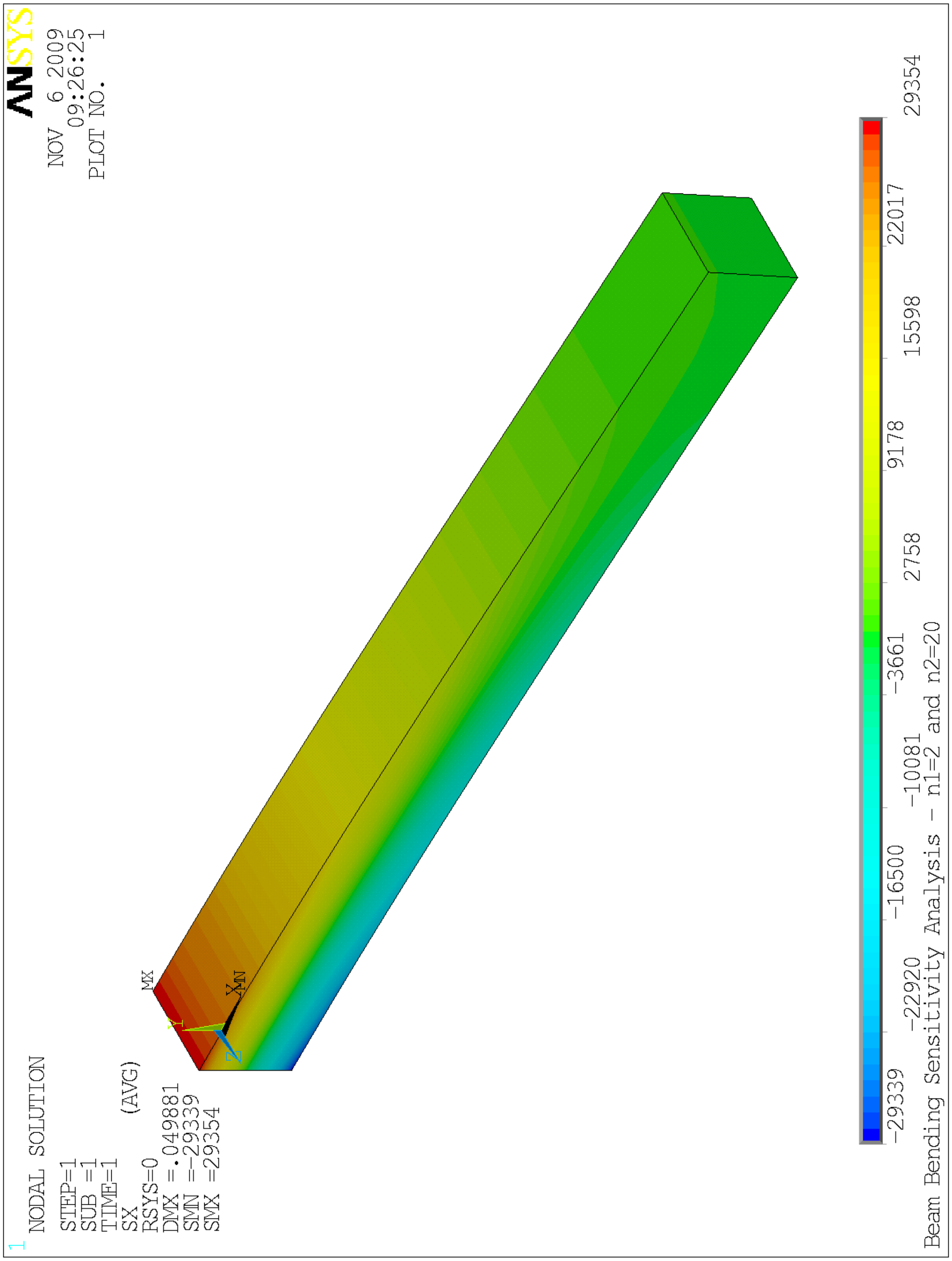



Figure-18

Beam Model Grid Sensitivity Study - A Typical Bending Stress Contour Plot

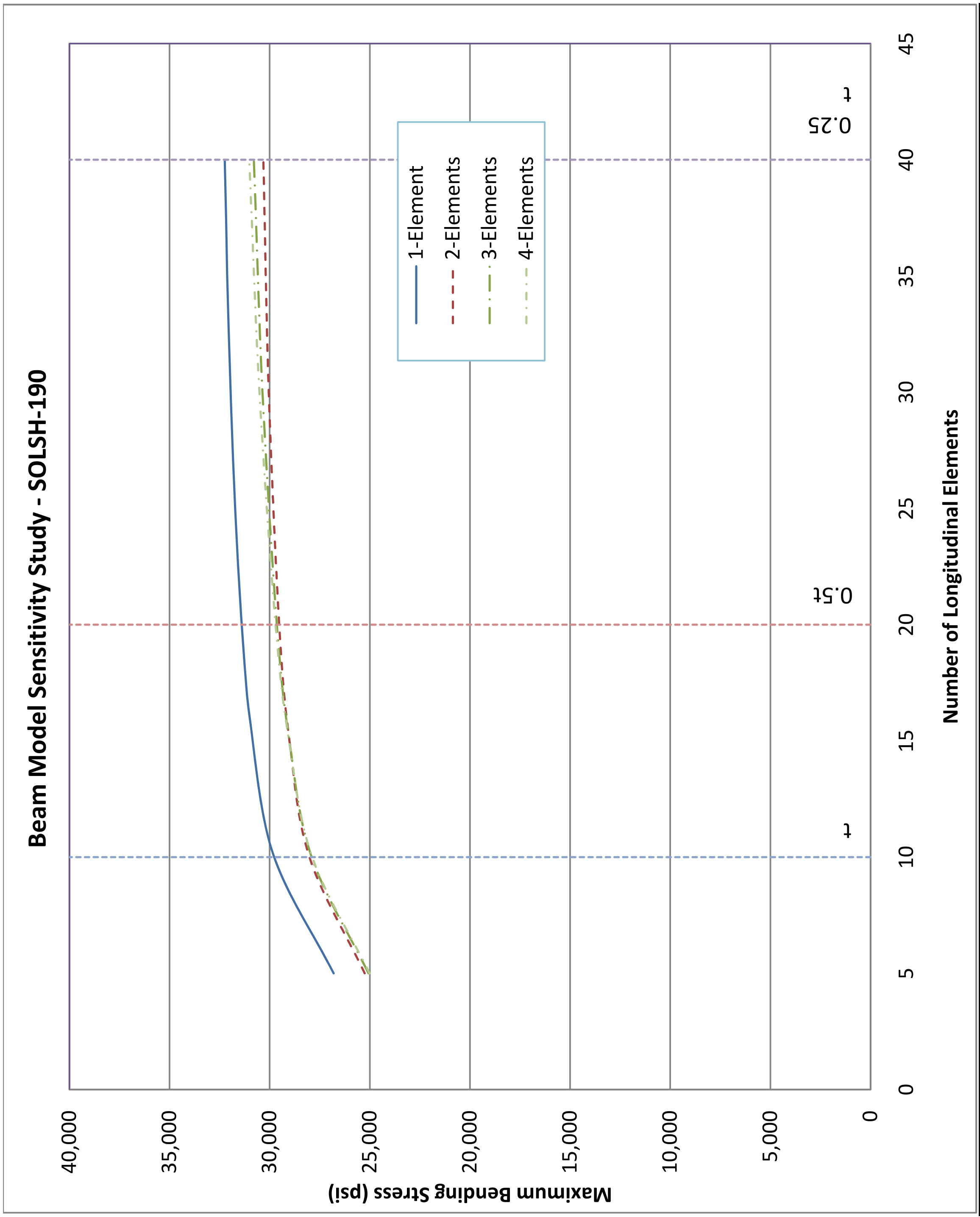



Figure-19

Beam Model Grid Sensitivity Study - Bending Stress Plot for Models with Various Grid Densities

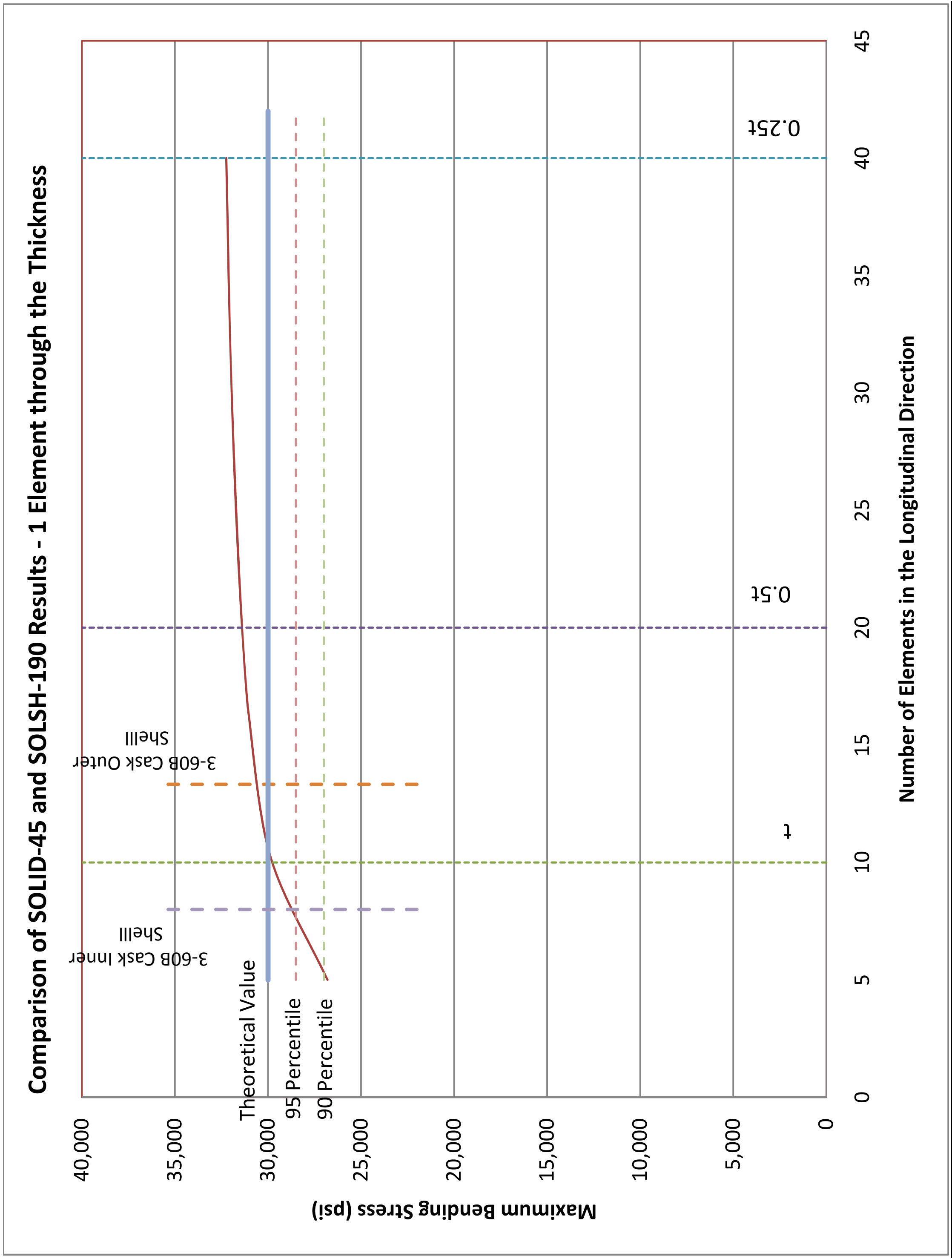



Figure-20

Beam Model Grid Sensitivity Study - Bending Stress Plot for Models with 1-Element through the Thickness

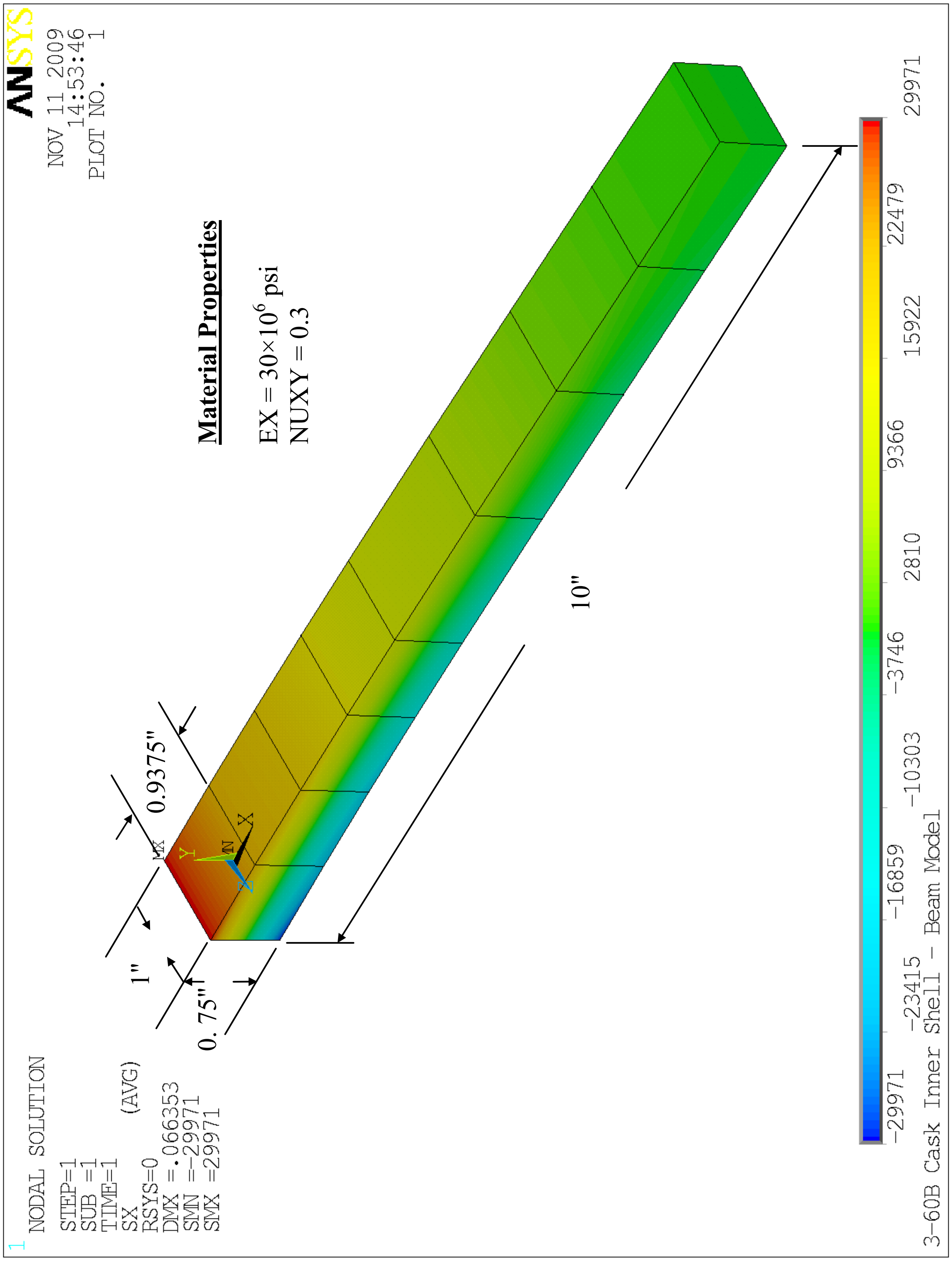



Figure-21

Beam Model Corresponding the SAR Inner Shell Modeling - Dimensions & Stress Contours

**Appendix 1**

**FEM Data Print-Out**

(8 Pages)

1. **<u>Partial Print-out of the SAR Model Database</u> (Reference Section 4.1.1)**

```
/COM,ANSYS RELEASE 11.0SP1 UP20070830       11:37:49    12/30/2009
/PREP7
/NOPR
/TITLE,3-60B Cask Bolting Ring Region - SAR Model
ANTYPE, 0
*IF,_CDRDOFF,EQ,1,THEN     !if solid model was read in
_CDRDOFF=             !reset flag, numoffs already performed
*ELSE              !offset database for the following FE model
NUMOFF,NODE,     490
NUMOFF,ELEM,     395
NUMOFF,MAT ,       1
NUMOFF,REAL,       4
NUMOFF,TYPE,       7
*ENDIF
*SET,I        ,  1.000000000000
*SET,MAXLAYER,  0.000000000000
*SET,_BUTTON ,  1.000000000000
*SET,_CMAP   ,  1.000000000000
*SET,_RETURN ,  0.000000000000
*SET,_RL1    ,  1.000000000000
*SET,_STATUS ,  1.000000000000
*SET,_UIQR   ,  1.000000000000
DOF,DELETE
ET,       2,185
ET,       3,190
ET,       4,170
ET,       5,175
KEYOP,       5, 9,        1
KEYOP,       5,10,        2
KEYOP,       5,12,        5
ET,       6,170
ET,       7,174
KEYOP,       7, 2,        2
KEYOP,       7, 4,        2
KEYOP,       7, 9,        1
KEYOP,       7,10,        2
KEYOP,       7,12,        5
```

The complete database is included in Appendix 2.

## 2. **Partial Print-out of the Fine Model Database** (Reference Section 4.1.2)

```
/COM,ANSYS RELEASE 11.0SP1 UP20070830       11:37:09    12/30/2009
/PREP7
/NOPR
/TITLE,3-60B Cask Bolting Ring Region - Fine Model with SOLID-185
ANTYPE, 0
*IF,_CDRDOFF,EQ,1,THEN     !if solid model was read in
_CDRDOFF=              !reset flag, numoffs already performed
*ELSE               !offset database for the following FE model
NUMOFF,NODE,    5562
NUMOFF,ELEM,    4288
NUMOFF,MAT ,       1
NUMOFF,TYPE,       1
NUMOFF,CSYS,       9
*ENDIF
*SET,I        ,  1.000000000000
*SET,MAXLAYER,  0.000000000000
*SET,R1       ,  17.50000000000
*SET,R2       ,  18.25000000000
*SET,R3       ,  18.62500000000
*SET,R4       ,  25.50000000000
*SET,Z1       ,  30.14700000000
*SET,Z2       ,  9.875000000000
*SET,Z3       ,  6.531200000000
*SET,Z4       ,  4.687500000000
*SET,_BUTTON ,  1.000000000000
*SET,_CHKMSH ,  0.000000000000
*SET,_CMAP   ,  1.000000000000
*SET,_RETURN ,  0.000000000000
*SET,_RL1    ,  1.000000000000
*SET,_STATUS ,  0.000000000000
*SET,_UIQR   ,  1.000000000000
DOF,DELETE
ET,       1,185
CSYS,   0
REAL,        2
```

The complete database is included in Appendix 2.

# 3. Input Data File for the Analyses of Section 4.2

## 3.1 Shell Model

```
C***   3-60B Cask Outer Shell - Inner Shell Grid Sensitivity Study
!
*dim,m,array,4,1
*dim,n,array,6,1
*dim,smbi,array,6,4
*dim,smbo,array,6,4
*dim,sii,array,6,4
*dim,sio,array,6,4
*dim,mom,array,6,4
m(1,1)=1
m(2,1)=2
m(3,1)=3
m(4,1)=4
n(1,1)=10
n(2,1)=15
n(3,1)=20
n(4,1)=25
n(5,1)=30
n(6,1)=40
*do,j,1,4,1
*do,i,1,6,1
/prep7
et,1,190
*get,stiff,etype,1,attr,enam
*use,steel,1
vclear,all
vdel,all,,,1
numcmp,all
csys,1
r1=17.5            ! Inside radius of the shell
r2=18.25           ! Outside radius of the shell
l=25               ! Shell length - arbitrary
t=10               ! 10-degee segment
n1=m(j,1)          ! segements along radial direction
n2=n(i,1)          ! segments along axial direction
n3=4               ! segments along tangential direction
/title,Inner Shell Grid Sensitivity Study - n1=%n1% and n2=%n2% - Element Type %stiff%
r=1                ! element size ratio
ir=1/r
t1=-t/2
k,1,r1,t1
k,2,r2,t1
k,3,r1,t1,l
k,4,r2,t1,l
l,1,2
l,2,4
l,4,3
l,3,1
al,1,2,3,4
```

```
lesize,1,,,n1
lesize,3,,,n1
lesize,2,,,n2,r
lesize,4,,,n2,ir
lgen,2,all,,,0,t
l,1,5
l,2,6
l,3,8
l,4,7
v,1,5,6,2,3,8,7,4
lesize,9,,,n3
lesize,10,,,n3
lesize,11,,,n3
lesize,12,,,n3
vsweep,1
/view,,1,1,1
/vup,,z
eplot
nsel,x,r1
sf,all,press,1000
nsel,z,0
d,all,uz
d,all,ux
nsel,y,t1
nasel,y,t1+t
nrot,all
d,all,uy
/solu
nall
eall
solve
/post1
set,last
/EDGE,1,1,45
/GLINE,1,0
/cont,1,64,auto
plnstr,sz
/show,term
PATH,sect,2,30,20,
n1=node(r1,0,0)
n2=node(r2,0,0)
PPATH,1,n1
PPATH,2,n2
prsect,,0
*get,smbi(i,j),section,sum,inside,s,z
*get,smbo(i,j),section,sum,outside,s,z
*get,sii(i,j),section,sum,inside,s,int
*get,sio(i,j),section,sum,outside,s,int
spoint,,17.875,0,0
nsel,z,0
fsum
*get,a,fsum,0,item,my
mom(i,j)=a/3.1198
```

```
*enddo            ! end of do loop i
*enddo            ! end of do loop j
*cfopen,et%stiff%.out
*vwrite,stiff
('Element Type',f10.0,' Results'/)
*vwrite
(//'Longitudinal Stress on the Inside Face'/)
*vwrite,m(1,1),m(2,1),m(3,1),m(4,1)
(5x,6f10.0)
*vwrite,n(1,1),smbi(1,1),smbi(1,2),smbi(1,3),smbi(1,4)
(f5.0,4f10.0)
*vwrite
(//'Longitudinal Stress on the Onside Face'/)
*vwrite,m(1,1),m(2,1),m(3,1),m(4,1)
(5x,4f10.0)
*vwrite,n(1,1),smbo(1,1),smbo(1,2),smbo(1,3),smbo(1,4)
(f5.0,4f10.0)
*vwrite
(//'Stress Intensity on the Inside Face'/)
*vwrite,m(1,1),m(2,1),m(3,1),m(4,1)
(5x,4f10.0)
*vwrite,n(1,1),sii(1,1),sii(1,2),sii(1,3),sii(1,4)
(f5.0,4f10.0)
*vwrite
(//'Stress Intensity on the Outside Face'/)
*vwrite,m(1,1),m(2,1),m(3,1),m(4,1)
(5x,4f10.0)
*vwrite,n(1,1),sio(1,1),sio(1,2),sio(1,3),sio(1,4)
(f5.0,4f10.0)
*vwrite
(//'End Moment per Inch'/)
*vwrite,m(1,1),m(2,1),m(3,1),m(4,1)
(5x,4f10.0)
*vwrite,n(1,1),mom(1,1),mom(1,2),mom(1,3),mom(1,4)
(f5.0,4f10.0)
*cfclose,et%stiff%.out
```

The electronic file of this data is included in Appendix 2.

## 3.2 Beam Model

```
C***   Beam Stress - Sensitivity Analysis
!
*dim,m,array,6,1
*dim,n,array,6,1
*dim,sigx,array,6,6
m(1,1)=1
m(2,1)=2
m(3,1)=3
m(4,1)=4
m(5,1)=6
m(6,1)=8
n(1,1)=5
n(2,1)=10
n(3,1)=15
n(4,1)=20
n(5,1)=30
n(6,1)=40
*do,j,2,2,1
*do,i,4,4,1
/prep7
et,1,185
*use,steel,1
vclear,all
vdel,all,,,1
numcmp,all
n1=m(j,1)          ! segements along depth direction
n2=n(i,1)          ! segments along length direction
n3=1               ! segments along width direction
/title,Beam Bending Sensitivity Analysis - n1=%n1% and n2=%n2%
r=1                ! element size ratio
ir=1/r
k,1,0,-0.5,-0.5
k,2,0,-0.5,0.5
k,3,0,0.5,-0.5
k,4,0,0.5,0.5
l,1,2
l,2,4
l,4,3
l,3,1
al,1,2,3,4
lesize,1,,,n3
lesize,3,,,n3
lesize,2,,,n1,r
lesize,4,,,n1,ir
lgen,2,all,,,10,0,0
l,1,5
l,2,6
l,3,8
l,4,7
v,1,5,6,2,3,8,7,4
lesize,9,,,n2
```

```
lesize,10,,,n2
lesize,11,,,n2
lesize,12,,,n2
vsweep,1
/view,,1,1,1
eplot
nsel,y,0.5
sf,all,press,100
nsel,x,0
d,all,ux,,,,,uy,uz
/solu
nall
eall
solve
/post1
set,last
/cont,1,64,auto
plnstr,sx
*get,a,plnsol,0,max
sigx(i,j)=a
/show,term
*enddo          ! end of do loop i
*enddo          ! end of do loop j
*get,stiff,etype,1,attr,enam
*cfopen,et%stiff%.out
*vwrite,stiff
('Element Type',f10.0,' Results'/)
*vwrite
(//'Maximum Bending Stress'/)
*vwrite,m(1,1),m(2,1),m(3,1),m(4,1),m(5,1),m(6,1)
(5x,6f10.0)
*vwrite,n(1,1),sigx(1,1),sigx(1,2),sigx(1,3),sigx(1,4),sigx(1,5),sigx(1,6)
(f5.0,6f10.0)
*cfclose,et%stiff%.out
```

The electronic file of this data is included in Appendix 2.

3.1 Partial Print-out of the Unit Width of the Inner Shell Model Database (Ref. Section 4.2.1)

```
/COM,ANSYS RELEASE 11.0SP1 UP20070830       13:22:24    12/30/2009
/PREP7
/NOPR
/TITLE,3-60B Cask Inner Shell - Beam Model
ANTYPE, 0
*IF,_CDRDOFF,EQ,1,THEN     !if solid model was read in
_CDRDOFF=             !reset flag, numoffs already performed
*ELSE              !offset database for the following FE model
NUMOFF,NODE,      39
NUMOFF,ELEM,       8
NUMOFF,MAT ,       1
NUMOFF,TYPE,       1
*ENDIF
*SET,_BUTTON ,  1.000000000000
*SET,_CMAP   ,  1.000000000000
*SET,_GUI_CLR_BG,' systemButtonFace              '
*SET,_GUI_CLR_FG,' systemButtonText              '
*SET,_GUI_CLR_INFOBG,' systemInfoBackground          '
*SET,_GUI_CLR_SEL,' systemHighlight               '
*SET,_GUI_CLR_SELBG,' systemHighlight               '
*SET,_GUI_CLR_SELFG,' systemHighlightText           '
*SET,_GUI_CLR_WIN,' systemWindow                  '
*SET,_GUI_FNT_FMLY,'Arial                          '
*SET,_GUI_FNT_PXLS,  16.00000000000
*SET,_GUI_FNT_SLNT,'r                              '
*SET,_GUI_FNT_WEGT,'medium                         '
*SET,_RETURN ,  0.000000000000
*SET,_RL1    ,  1.000000000000
*SET,_STATUS ,  1.000000000000
*SET,_UIQR   ,  0.000000000000
DOF,DELETE
ET,       1,190
```

The complete database is included in Appendix 2.

## Appendix 2

## FEM Electronic Data

(2 Pages + 1 CD)

## Directory of Files on the CD

```
 Volume in drive F is ST-608 Rev.0 App
 Volume Serial Number is EA86-CE64

 Directory of F:\

12/18/2009  11:53 AM    <DIR>          Joint Evaluation
12/30/2009  03:11 PM    <DIR>          Shell Evaluation
               0 File(s)              0 bytes

 Directory of F:\Joint Evaluation

12/18/2009  11:53 AM    <DIR>          .
12/30/2009  03:35 PM    <DIR>          ..
12/30/2009  03:10 PM    <DIR>          Fine Model
12/30/2009  03:11 PM    <DIR>          SAR Model
               0 File(s)              0 bytes

 Directory of F:\Joint Evaluation\Fine Model

12/30/2009  03:10 PM    <DIR>          .
12/18/2009  11:53 AM    <DIR>          ..
12/30/2009  11:37 AM         1,305,837 file.cdb
12/23/2009  03:45 PM        17,170,432 file.db
12/01/2009  01:52 PM         8,257,536 file.rst
11/30/2009  04:17 PM           133,291 file000.png
11/30/2009  04:27 PM           107,324 file001.png
11/30/2009  04:28 PM            71,515 file002.png
11/30/2009  04:29 PM            28,878 file003.png
11/30/2009  04:29 PM            30,203 file004.png
12/02/2009  03:40 PM            59,765 file006.png
12/30/2009  11:25 AM           958,598 model.out
              10 File(s)     28,123,379 bytes

 Directory of F:\Joint Evaluation\SAR Model

12/30/2009  03:11 PM    <DIR>          .
12/18/2009  11:53 AM    <DIR>          ..
12/30/2009  11:37 AM           116,229 file.cdb
12/03/2009  11:05 AM         2,621,440 file.db
12/01/2009  04:49 PM           917,504 file.rst
11/30/2009  05:35 PM            26,677 file000.png
11/30/2009  05:35 PM            27,049 file001.png
12/01/2009  04:52 PM            62,182 file002.png
12/02/2009  03:39 PM            85,865 file004.png
11/30/2009  05:12 PM            58,391 file005.png
11/30/2009  05:14 PM           109,167 file007.png
11/30/2009  05:14 PM            71,049 file008.png
12/01/2009  04:03 PM            51,985 model.png
              11 File(s)      4,147,538 bytes
```

```
 Directory of F:\Shell Evaluation

12/30/2009  03:11 PM    <DIR>          .
12/30/2009  03:35 PM    <DIR>          ..
12/30/2009  03:30 PM    <DIR>          Beam
12/23/2009  08:33 AM             1,885 et190.out
12/23/2009  09:17 AM         3,932,160 file.db
12/23/2009  08:33 AM         1,835,008 file.rst
12/21/2009  10:03 AM            32,391 file000.png
12/21/2009  10:05 AM            57,237 file001.png
12/21/2009  10:05 AM            57,223 file002.png
12/23/2009  08:32 AM             2,740 Input.txt
12/21/2009  01:22 PM            14,732 LongStress.xlsx
12/30/2009  11:04 AM            14,897 Moment.xlsx
               9 File(s)      5,948,273 bytes

 Directory of F:\Shell Evaluation\Beam

12/30/2009  03:30 PM    <DIR>          .
12/30/2009  03:11 PM    <DIR>          ..
12/21/2009  03:21 PM            21,614 Beam Sensitivity.xlsx
12/30/2009  03:29 PM               533 et190.out
12/30/2009  03:29 PM         2,686,976 file.db
12/30/2009  03:29 PM         1,114,112 file.rst
11/06/2009  09:25 AM            42,713 file000.png
11/06/2009  09:26 AM            67,732 file001.png
12/30/2009  03:30 PM    <DIR>          Inner Shell
12/30/2009  03:28 PM             1,457 Input.txt
               7 File(s)      3,935,137 bytes

 Directory of F:\Shell Evaluation\Beam\Inner Shell

12/30/2009  03:30 PM    <DIR>          .
12/30/2009  03:30 PM    <DIR>          ..
12/30/2009  01:22 PM             8,013 file.cdb
11/11/2009  03:13 PM         1,376,256 file.db
11/11/2009  02:51 PM           327,680 file.rst
11/11/2009  02:53 PM            84,126 file001.png
12/30/2009  01:21 PM            17,733 model.out
               5 File(s)      1,813,808 bytes

     Total Files Listed:
              42 File(s)     43,968,135 bytes
              18 Dir(s)               0 bytes free
```