

1.0 OBJECTIVE

Calculation of stiffnesses of the 8-120B Cask wall components under concentrated load.

2.0 REFERENCES

1. EnergySolutions Document ST-627, Rev.1, Structural Analyses of the 8-120B Cask under Drop Conditions.
2. Roark and Young, *Formulas for Stress and Strain*, Fifth Edition, McGraw-Hill Book Company, 1975.
3. ANSYS, Release 13.0, ANSYS Inc., Canonsburg, PA, 2010.

3.0 INTRODUCTION

In the EnergySolutions document ST-627 (Reference 1) the 8-120B Cask was evaluated under various drop conditions. Under the puncture drop analysis, the lead shield deformation was calculated using closed-form solutions from Reference 2. An assumption that was made in the calculation needs further justification, which is presented in this document.

Formulas for the deflection under a concentrated load on a long cylinder were used to calculate the stiffness of each component of the cask wall – outer shell, lead-shield, and the inner shell. The equivalent stiffness of the wall was calculated by summing up the individual stiffnesses. Since the material properties of steel and lead are considerably different and the lead shielding is not chemically bonded to the shell surface, this assumption needs further justifications.

ANSYS (Reference 3) finite element models have been used to provide this justification. Stiffnesses of the individual components, and the composite section, have been calculated and compared with the theoretical values.

The finite element models of the three components of the 8-120B Cask are constructed using 3-dimensional solid elements. The interfaces between the outer shell and lead-shielding and between the inner shell and the lead-shielding are constructed from 3-dimensional contact elements. These elements prevent penetration but allow sliding of one surface with respect to another surface. An arbitrary concentrated load of 5,000 lb is applied to the half model and the resulting deflection is used for the stiffness calculation.

The results of the analyses show the assumption used in the equivalent stiffness calculation in Reference 1 is totally justified.

4.0 FINITE ELEMENT MODEL DESCRIPTION

The finite element model consists of 3-dimensional structural solid (SOLID 45) elements to represent the steel shells and lead-shielding, 3-dimensional contact (CONTA174) and target (TARGE170) pairs to represent the interfaces between the shells and the lead-shielding. The material properties of steel and lead are defined by linear isotropic models. Because of the symmetry only ½ of the wall components have been modeled. The finite element model is shown in Figure 1. Figure 2 shows the interface elements used in the model. Figure 3 shows the boundary conditions employed. Appendix 1 gives the print-out of the finite element model.

A concentrated load of 5,000 lb is applied at the mid-height of the wall components and the displacement under this load is used to calculate the stiffness. Appendix 2 gives the analyses results.

5.0 ANALYSES RESULTS

The model is analyzed in four load steps. In load step 1, the outer shell half model is subjected to the concentrated load of 5,000 lb. all other elements are voided. In steps 2 and 3 the same process is done for the lead-shielding and the inner shell, respectively. In step 4 all the components together are subjected to the same concentrated load.

5.1 OUTER SHELL STIFFNESS

The outer shell stiffness under a concentrated load is calculated from the results of load step 1. Please refer to Appendix 2 and Figure 4 for the result output.

Applied load on the half model	= 5,000 lb
Displacement under the load	= 0.0060321 in
Stiffness, k_1	= $2 \times 5,000 / 0.0060321 = 1.658 \times 10^6$ lb/in

It should be noted that the stiffness calculated in Reference 1, using the formulas from Reference 2 is 1.294×10^6 lb/in, which is slightly smaller than the one calculated using the finite element model. This is expected since the finite size of the elements in the FEM under concentrated load predicts lower displacement and hence higher stiffness. Since the objective of this document is to validate the assumption used in calculating the equivalent stiffness, this variation is of secondary importance.

5.2 LEAD-SHIELD STIFFNESS

The lead-shield stiffness under a concentrated load is calculated from the results of load step 2. Please refer to Appendix 2 and Figure 5 for the result output.

Applied load on the half model	= 5,000 lb
Displacement under the load	= 0.013943 in
Stiffness, k_2	= $2 \times 5,000 / 0.013943 = 717,206$ lb/in

5.3 INNER SHELL STIFFNESS

The inner shell stiffness under a concentrated load is calculated from the results of load step 3. Please refer to Appendix 2 and Figure 6 for the result output.

Applied load on the half model	= 5,000 lb
Displacement under the load	= 0.024309 in
Stiffness, k_3	= $2 \times 5,000 / 0.024309 = 411,370$ lb/in

5.4 WALL STIFFNESS

The wall stiffness under a concentrated load is calculated from the results of load step 4. Please refer to Appendix 2 and Figure 7 for the result output.

Applied load on the half model	= 5,000 lb
--------------------------------	------------

Displacement under the load = 0.003524 in

Stiffness, k_{eqv} = $2 \times 5,000 / 0.003524 = 2.838 \times 10^6$ lb/in

5.5 EQUIVALENT STIFFNESS

Based on the assumption that the stiffnesses of the individual components can be added to obtain the equivalent stiffness of the wall, we get:

$$k_1 + k_2 + k_3 = 1.658 \times 10^6 + 717,206 + 411,370 = 2.787 \times 10^6 \text{ lb/in}$$

Thus the equivalent stiffness calculated by the summation is within 1.8% of the computed value. Therefore, it is concluded that the assumption gives the valid result on the equivalent stiffness.

5.0 CONCLUSIONS

Although the relative stiffnesses of the steel and lead materials, and the un-bonded nature of the lead-steel interfaces have some effect on the stiffness of the composite section, based on the geometry of the 8-120B Cask inner and outer shells and the lead-shielding, it is shown that the equivalent stiffness of the wall can be calculated by summing the individual stiffnesses of the wall components. Therefore, the assumption used in the calculations of Reference 1 is validated by the FEM analyses provided in this document.

Title Calculation of Stifnesses of the 8-120B Cask Wall Components under Concentrated Loads

Calc. No. ST-679 **Rev.** 0

Sheet 4 **of** 7

Figures

(7 Pages)

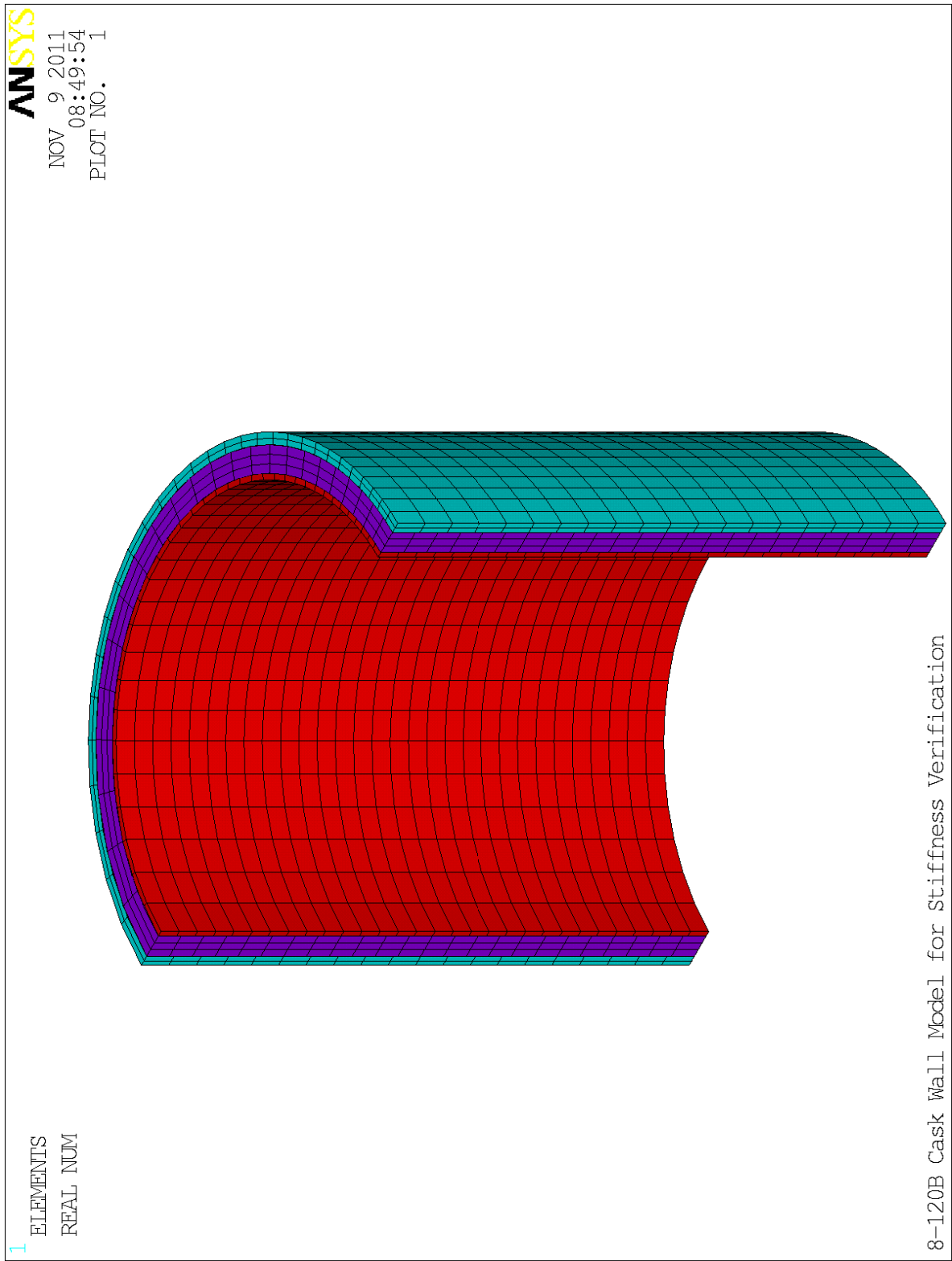


Figure 1 – 8-120B Cask Wall Finite Element Model

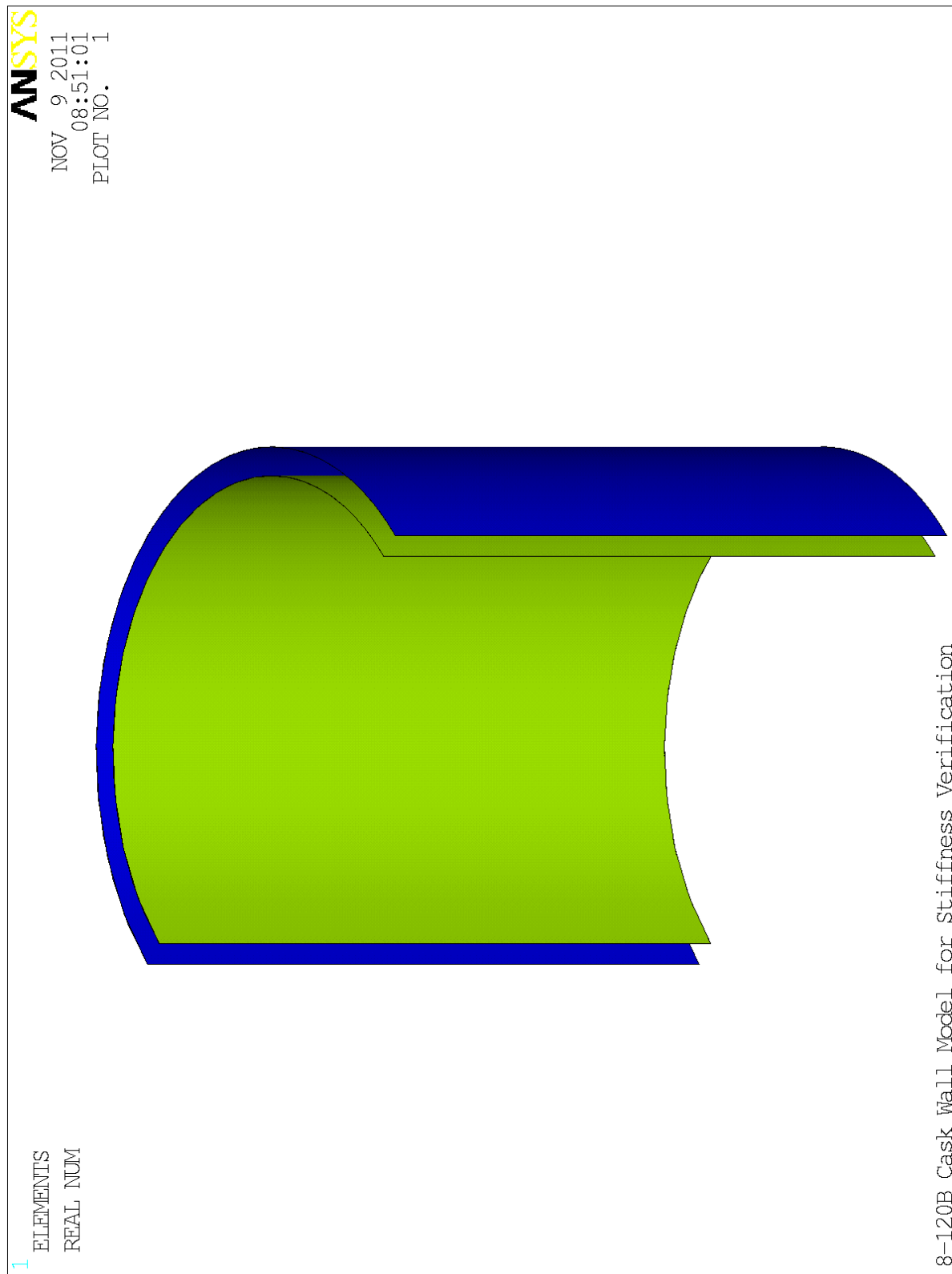


Figure 2 – Contact Elements Used in the Finite Element Model

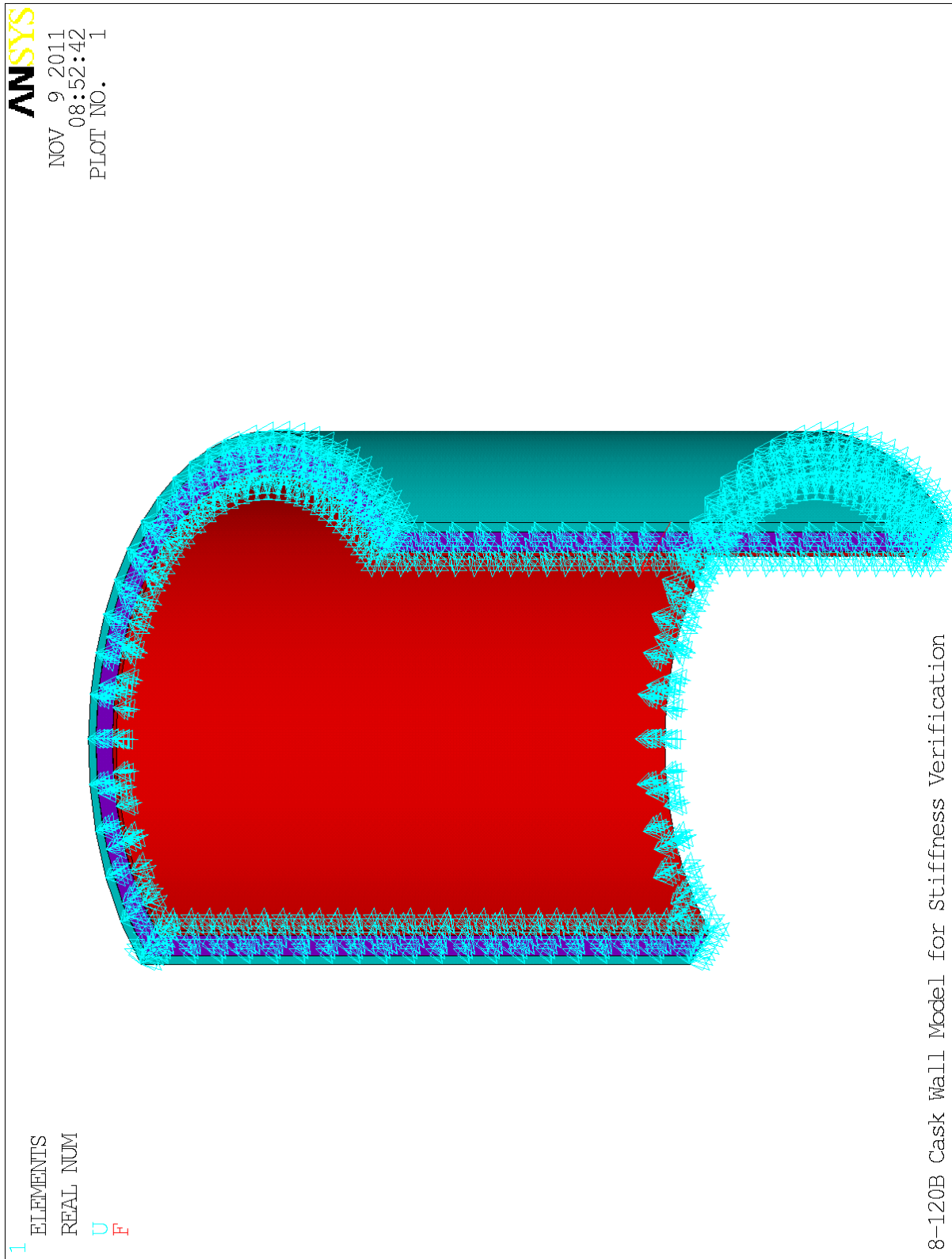


Figure 3 – Boundary Conditions Used in the Finite Element Model

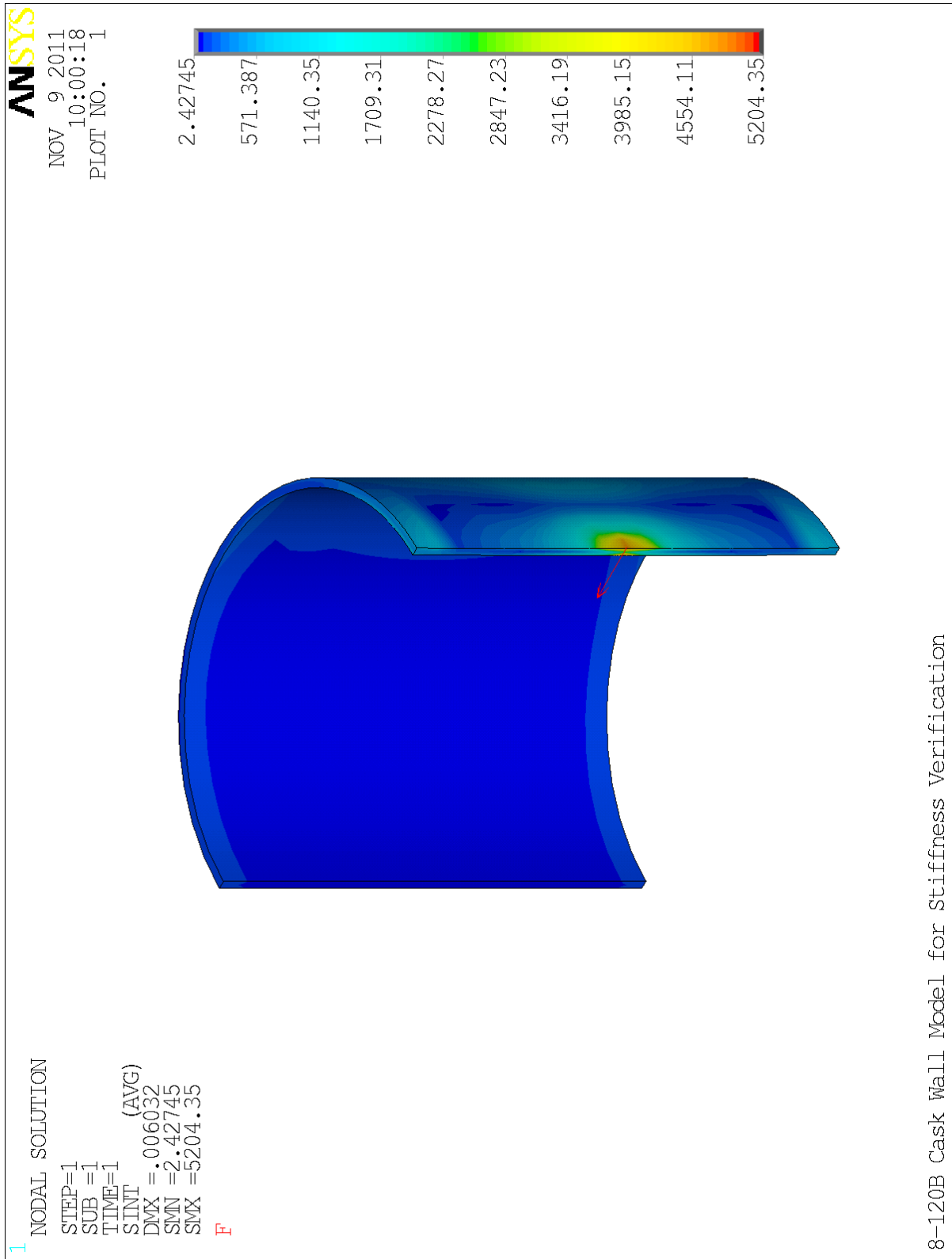


Figure 4 – Outer Shell under Applied Concentrated Load of 5,000 lb

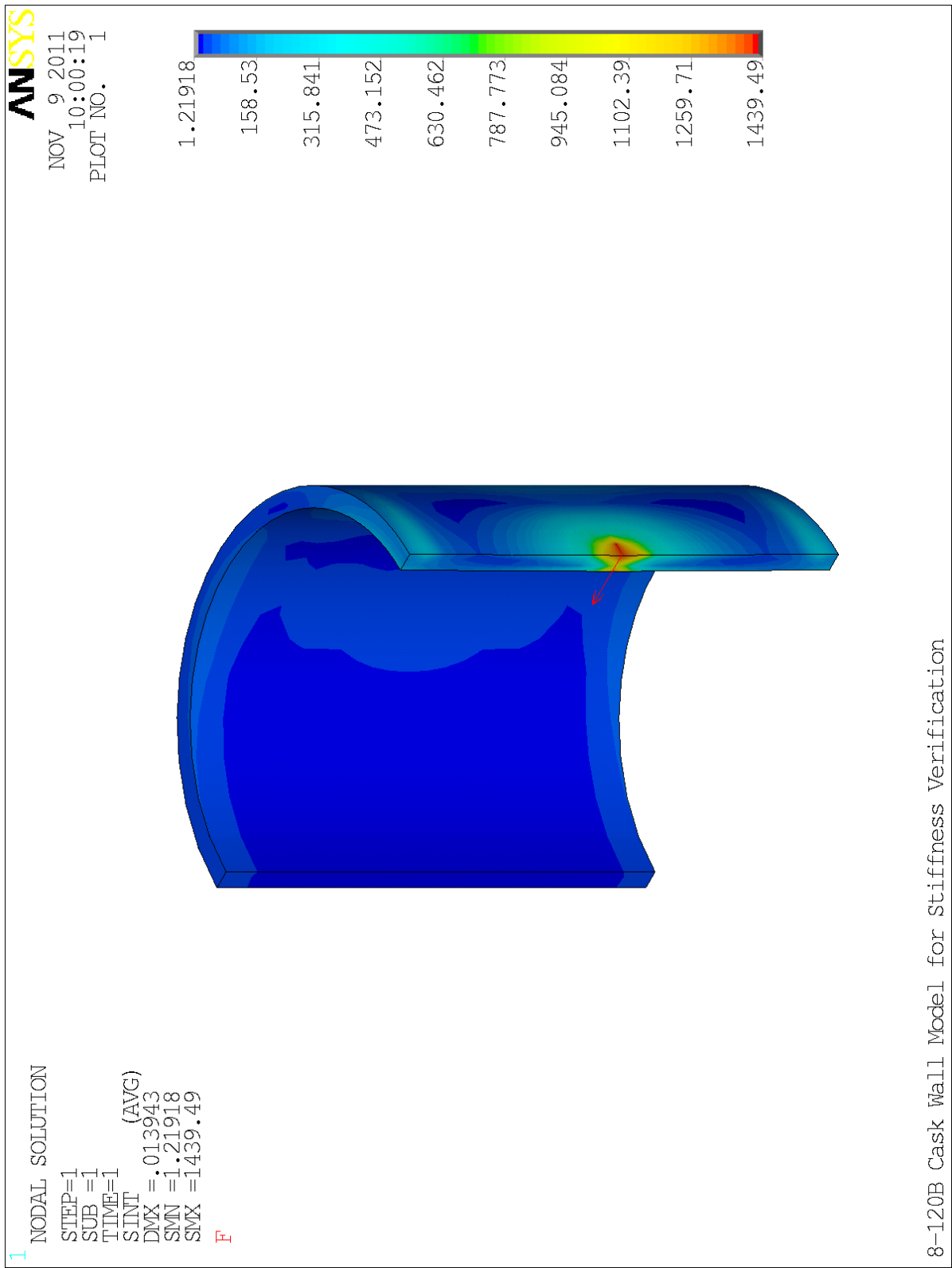


Figure 5 – Lead Shielding under Applied Concentrated Load of 5,000 lb

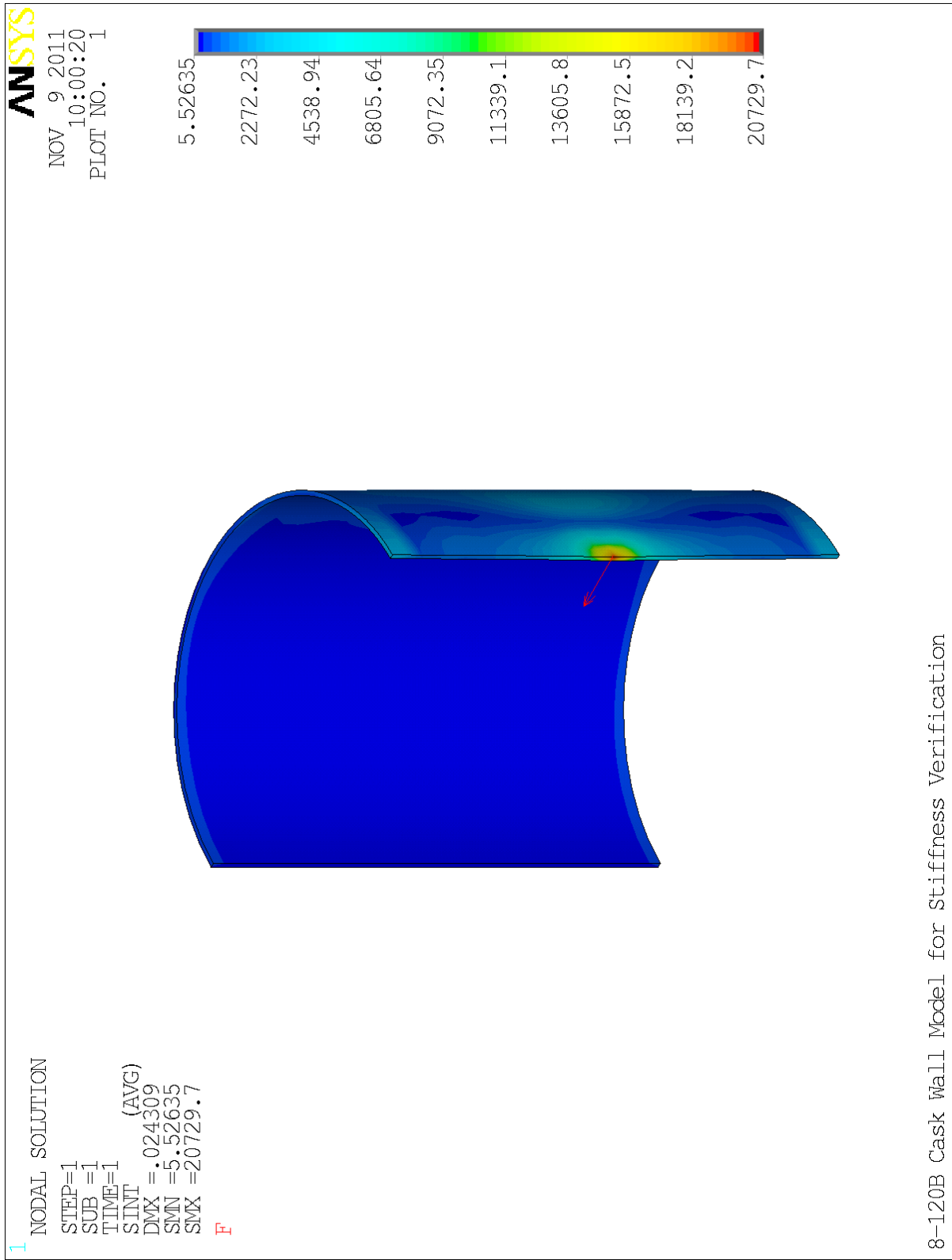


Figure 6 – Inner Shell under Applied Concentrated Load of 5,000 lb

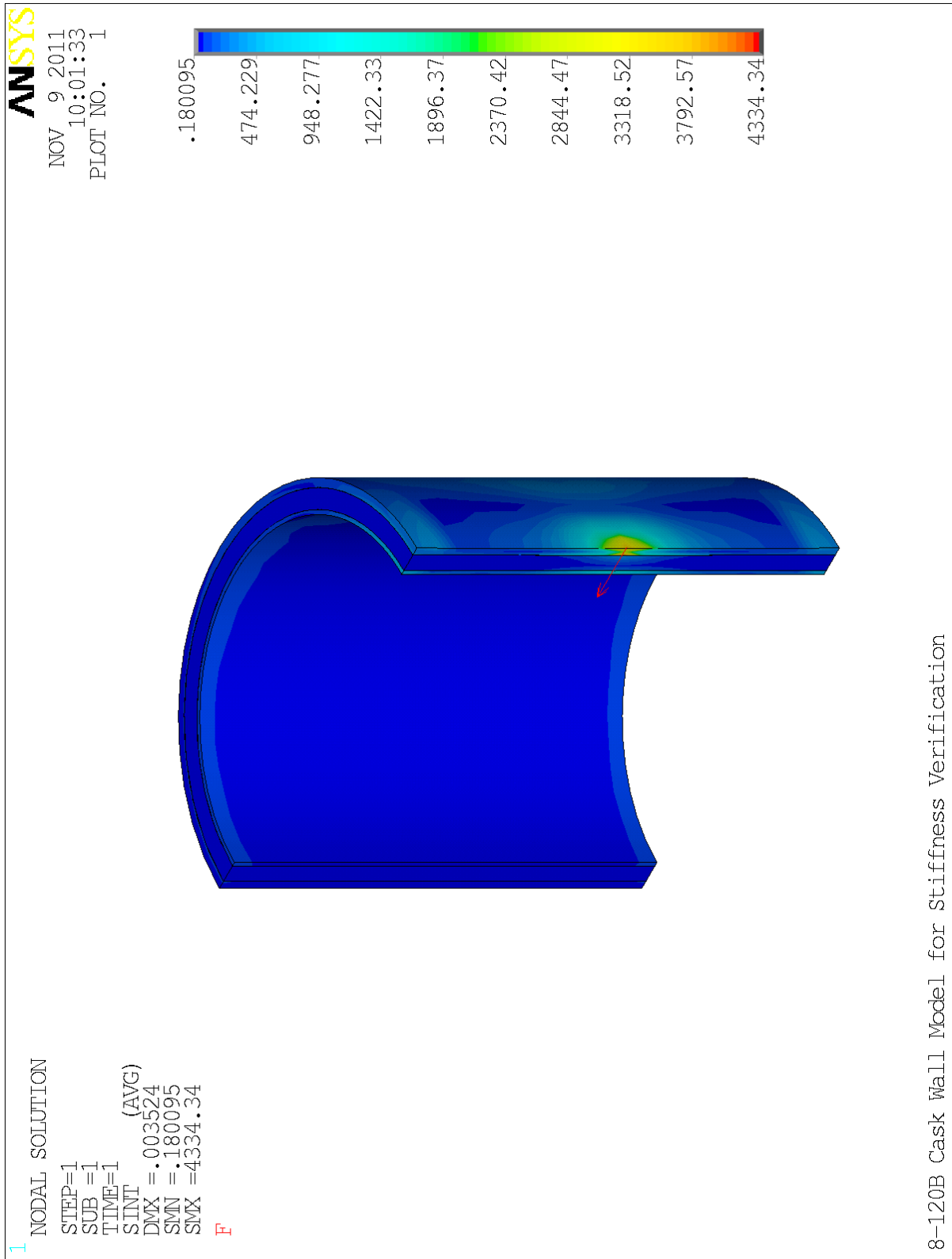


Figure 7 – 8-120B Composite Wall under Applied Concentrated Load of 5,000 lb

Title Calculation of Stifnesses of the 8-120B Cask Wall Components under Concentrated Loads

Calc. No. ST-679 **Rev.** 0

Sheet 5 **of** 7

APPENDIX 1

(6 Pages)

ANSYS Finite Element Model Listing

G L O B A L S T A T U S

ANSYS - Engineering Analysis System Nov 09, 2011 10:04
 Release 13.0 00222442 WINDOWS x64 Version

Current working directory: D:\ANSYS Analyses\Dummy\8-120B Puncture

MENULIST File: C:\Program Files\ANSYS Inc\v130\ANSYS\gui\en-us\UIDL\menulist130.ans

Product(s) enabled: ANSYS Mechanical

Total connect time. 1 hours 27 minutes
 Total CP usage. 0 hours 4 minutes 12.6 seconds

J O B I N F O R M A T I O N -----

8-120B Cask Wall Model for Stiffness Verification

Current jobnamefile
 Initial jobnamefile

Unitsunknown

	Available	Used
Scratch Memory Space.	9600.000 mb	532.599 mb (5.5%)
Database space	65535.750 mb	26.596 mb (0.0%)

User menu file in use . . . %ANSYS130_DIR%\gui\en-us\UIDL\UIMENU.GRN
 User menu file in use . . . %ANSYS130_DIR%\gui\en-us\UIDL\UIFUNC1.GRN
 User menu file in use . . . %ANSYS130_DIR%\gui\en-us\UIDL\UIFUNC2.GRN
 User menu file in use . . . %ANSYS130_DIR%\gui\en-us\UIDL\MECHTOOL.AUI
 Beta features are not shown in the user interface

M O D E L I N F O R M A T I O N -----

Solid model summary:

	Largest Number	Number Defined	Number Selected
Keypoints	24	24	8
Lines	36	36	12
Areas	18	18	6
Volumes	3	3	1

Finite element model summary:

	Largest Number	Number Defined	Number Selected
Nodes	6747	6747	6747
Elements.	15816	6840	6840

Element types	17	9	n.a.
Real constant sets.	10	4	n.a.
Material property sets. . .	2	2	n.a.
Coupling.	0	0	n.a.
Constraint equations. . . .	0	0	n.a.
Master DOFs	0	0	n.a.
Dynamic gap conditions. . .	0	0	n.a.

BOUNDARY CONDITION INFORMATION -----

	Number Defined			
Constraints on nodes.	1518			
Constraints on keypoints.	0			
Constraints on lines.	0			
Constraints on areas.	0			
Forces on nodes	1			
Forces on keypoints	0			
Surface loads on elements	0			
Number of element flagged surfaces . . .	0			
Surface loads on lines.	0			
Surface loads on areas.	0			
Body loads on elements.	0			
Body loads on areas	0			
Body loads on lines	0			
Body loads on nodes	0			
Body loads on keypoints	0			
Temperatures				
Uniform temperature.	0.000			
Reference temperature.	0.000			
Offset from absolute scale	0.000			
	X	Y	Z	
Linear acceleration	0.0000	0.0000	0.0000	
Angular velocity (about global CS). . . .	0.0000	0.0000	0.0000	
Angular acceleration (about global CS). .	0.0000	0.0000	0.0000	
Location of reference CS.	0.0000	0.0000	0.0000	
Angular velocity (about reference CS) . .	0.0000	0.0000	0.0000	
Angular acceleration (about reference CS)	0.0000	0.0000	0.0000	

ROUTINE INFORMATION -----

Current routine.Preprocessing (PREP7)

Active coordinate system 11 (Cylindrical)

Display coordinate system. 0 (Cartesian)

Current element attributes:

Type number	17	(CONTA174)
Real number	10	
Material number	1	

```

Element coordinate system number. . . . . 0

Current mesher type. . . . . .based on default element shape

Current element meshing shape 2D . . .use default element shape.

Current element meshing shape 3D . . .use default element shape.

SmrtSize Level . . . . . OFF

Global element size. . . . . 0 divisions per line

Active coordinate system . . . . . 11 (Cylindrical)

Display coordinate system. . . . . 0 (Cartesian)

Analysis type. . . . . .Static (steady-state)

Active options for this analysis type:
  Large deformation effects . . . . .Not included
  Plasticity. . . . . .Not included
  Creep . . . . . .Not included
  Equation solver to use. . . . . .Program Chosen

Results file . . . . . .file.rst

Load step number . . . . . 2

Number of substeps . . . . . 1
  Step change boundary conditions . .No

```

Analysis Options

```

New, Restart, or Expansion Pass: NEW ANALYSIS
Discipline (based on active DOF): STRUCTURAL
Analysis type: STATIC

Prestress or stress stiff. effects NO PRESTRES/STRESS STIFF
Large deformation effects DON'T INCLUDE LG DEFORM

Newton-Raphson option PROGRAM CHOOSES
Newton-Raphson adaptive descent DO NOT USE ADAPT DESCENT

Mass Matrix formulation DEFAULT ELEM MASS MATRIX
Equation solver to be used PROGRAM CHOOSES
Iterative Solver tolerance value 0.10000E-07

Difference (in degrees) between absolute zero and
the temperature system being used 0.0000

LIST ELEMENT TYPES FROM 1 TO 17 BY 1

ELEMENT TYPE 1 IS SOLID45 3-D STRUCTURAL SOLID
KEYOPT( 1- 6)= 0 0 0 0 0 0

```

```

KEYOPT( 7-12)=          0      0      0      0      0      0
KEYOPT(13-18)=          0      0      0      0      0      0

ELEMENT TYPE          6 IS TARGE170      3-D TARGET SEGMENT
KEYOPT( 1- 6)=          0      0      0      0      0      0
KEYOPT( 7-12)=          0      0      0      0      0      0
KEYOPT(13-18)=          0      0      0      0      0      0

ELEMENT TYPE          7 IS CONTA174      3D 8-NODE SURF-SURF CONTACT
KEYOPT( 1- 6)=          0      0      0      0      3      0
KEYOPT( 7-12)=          0      0      1      2      0      0
KEYOPT(13-18)=          0      0      0      0      0      0

ELEMENT TYPE          8 IS TARGE170      3-D TARGET SEGMENT
KEYOPT( 1- 6)=          0      0      0      0      0      0
KEYOPT( 7-12)=          0      0      0      0      0      0
KEYOPT(13-18)=          0      0      0      0      0      0

ELEMENT TYPE          9 IS CONTA174      3D 8-NODE SURF-SURF CONTACT
KEYOPT( 1- 6)=          0      0      0      0      3      0
KEYOPT( 7-12)=          0      0      1      2      0      0
KEYOPT(13-18)=          0      0      0      0      0      0

ELEMENT TYPE         14 IS TARGE170      3-D TARGET SEGMENT
KEYOPT( 1- 6)=          0      0      0      0      0      0
KEYOPT( 7-12)=          0      0      0      0      0      0
KEYOPT(13-18)=          0      0      0      0      0      0

ELEMENT TYPE         15 IS CONTA174      3D 8-NODE SURF-SURF CONTACT
KEYOPT( 1- 6)=          0      0      0      0      0      0
KEYOPT( 7-12)=          0      0      1      2      0      0
KEYOPT(13-18)=          0      0      0      0      0      0

ELEMENT TYPE         16 IS TARGE170      3-D TARGET SEGMENT
KEYOPT( 1- 6)=          0      0      0      0      0      0
KEYOPT( 7-12)=          0      0      0      0      0      0
KEYOPT(13-18)=          0      0      0      0      0      0

ELEMENT TYPE         17 IS CONTA174      3D 8-NODE SURF-SURF CONTACT
KEYOPT( 1- 6)=          0      0      0      0      0      0
KEYOPT( 7-12)=          0      0      1      2      0      0
KEYOPT(13-18)=          0      0      0      0      0      0

CURRENT NODAL DOF SET IS  UX      UY      UZ
THREE-DIMENSIONAL MODEL

LIST MATERIALS          1 TO          2 BY          1
PROPERTY= ALL

MATERIAL NUMBER          1

      TEMP          EX
      0.3000000E+08

      TEMP          NUXY
      0.3000000

```


TEMP	DENS	
	0.2830000	
TEMP	MU	
	0.000000	
TEMP	EMIS	
	0.7888609E-30	
MATERIAL NUMBER		2
TEMP	EX	
-40.000	0.24600E+07	
-20.000	0.24300E+07	
70.000	0.22700E+07	
100.00	0.22100E+07	
200.00	0.20100E+07	
300.00	0.18500E+07	
400.00	0.17000E+07	
500.00	0.15200E+07	
TEMP	NUXY	
	0.4000000	
TEMP	ALPX	REFERENCE TEMP. = 0.00
-40.000	0.15560E-04	
-20.000	0.15650E-04	
70.000	0.16060E-04	
100.00	0.16220E-04	
200.00	0.16700E-04	
300.00	0.17330E-04	
400.00	0.18160E-04	
500.00	0.19120E-04	
TEMP	DENS	
	0.4100000	

General load step options

Automatic time stepping	USE AUTOMATIC TIME STEPPING
Number of substeps	1
Time at end of load step	TIME = 0.0000
Stiffness matrix reuse options	PROGRAM DECIDES
Reference temperature	TREF= 0.000
*** NOTE ***	CP = 252.706 TIME= 10:04:55
No nodal body forces to list.	

Inertia load options

Acceleration vector	GLOBAL CARTESIAN COMPONENTS ARE:
	0.0000 0.0000 0.0000

```

Angular velocity vector      GLOBAL CARTESIAN COMPONENTS ARE:
                             0.0000      0.0000      0.0000
                             SPIN SOFTENING NOT ACTIVATED
Angular acceleration vector  GLOBAL CARTESIAN COMPONENTS ARE:
                             0.0000      0.0000      0.0000
Reference coord. system origin ORIGIN = 0.0000      0.0000      0.0000
Angular velocity vector      REFERENCE COORDINATE COMPONENTS ARE:
                             0.0000      0.0000      0.0000
Angular acceleration vector  REFERENCE COORDINATE COMPONENTS ARE:
                             0.0000      0.0000      0.0000
Inertia relief              NO INERTIA RELIEF
Translational acceleration vector on components  NONE
Angular velocity vector on components  NONE
Angular acceleration vector on components  NONE

```

```

LIST NODAL FORCES FOR SELECTED NODES      1 TO      6747 BY      1
CURRENTLY SELECTED NODAL LOAD SET= FX      FY      FZ

```

NODE	LABEL	REAL	IMAG
813	FX	-5000.00000	0.00000000

Title Calculation of Stifnesses of the 8-120B Cask Wall Components under Concentrated Loads

Calc. No. ST-679 **Rev.** 0

Sheet 6 **of** 7

APPENDIX 2

(8 Pages)

Analysis Result Print-Out

***** ANSYS SOLUTION ROUTINE *****

c* Outer shell stiffness**

```

ESEL  FOR LABEL= REAL  FROM          1 TO          1 BY          1
      1200  ELEMENTS (OF          6840  DEFINED) SELECTED BY  ESEL  COMMAND.
ERSE  FOR LABEL= TYPE  FROM          1 TO          1 BY          1
      1200  ELEMENTS (OF          6840  DEFINED) SELECTED BY  ERSE  COMMAND.
SELECT      ALL NODES HAVING ANY ELEMENT IN ELEMENT SET.

      1953  NODES (OF          6747  DEFINED) SELECTED FROM
      1200  SELECTED ELEMENTS BY NELE COMMAND.

DELETE ALL SPECIFIED NODAL LOADS FROM NODE          1 TO          6747 BY          1
NUMBER OF NODAL LOADS DELETED=          1

SPECIFIED NODAL LOAD FX  FOR SELECTED NODES          813 TO          813 BY          1
REAL= -5000.00000      IMAG=  0.000000000

```

***** ANSYS SOLVE COMMAND *****

S O L U T I O N O P T I O N S

```

PROBLEM DIMENSIONALITY. . . . . .3-D
DEGREES OF FREEDOM. . . . . UX  UY  UZ
ANALYSIS TYPE . . . . . .STATIC (STEADY-STATE)
NEWTON-RAPHSON OPTION . . . . . .PROGRAM CHOSEN
GLOBALLY ASSEMBLED MATRIX . . . . . .SYMMETRIC

```

L O A D S T E P O P T I O N S

```

LOAD STEP NUMBER. . . . . 1
TIME AT END OF THE LOAD STEP. . . . . 1.0000
NUMBER OF SUBSTEPS. . . . . 1
MAXIMUM NUMBER OF EQUILIBRIUM ITERATIONS. . . . 15
STEP CHANGE BOUNDARY CONDITIONS . . . . . NO
TERMINATE ANALYSIS IF NOT CONVERGED . . . . .YES (EXIT)
CONVERGENCE CONTROLS. . . . . .USE DEFAULTS
PRINT OUTPUT CONTROLS . . . . . .NO PRINTOUT
DATABASE OUTPUT CONTROLS. . . . . .ALL DATA WRITTEN
                                     FOR THE LAST SUBSTEP

```

SOLUTION MONITORING INFO IS WRITTEN TO FILE= file.mntr

```

Element Formation Element= 1000 Cum.  Iter.= 1 CP= 214.720
Time= 1.0000 Load Step= 1 Substep= 1 Equilibrium Iteration= 1.

```

**** CENTER OF MASS, MASS, AND MASS MOMENTS OF INERTIA ****

CALCULATIONS ASSUME ELEMENT MASS AT ELEMENT CENTROID

TOTAL MASS = 3732.0

CENTER OF MASS	MOM. OF INERTIA ABOUT ORIGIN	MOM. OF INERTIA ABOUT CENTER OF MASS
XC = 0.49958E-01	IXX = 0.1000E+08	IXX = 0.2350E+07
YC = 39.125	IYY = 0.4781E+07	IYY = 0.2840E+07
ZC = -22.800	IZZ = 0.1000E+08	IZZ = 0.4289E+07
	IXY = -7295.	IXY = 0.4574E-08
	IYZ = 0.3329E+07	IYZ = -0.1630E-07
	IZX = 2631.	IZX = -1620.

*** MASS SUMMARY BY ELEMENT TYPE ***

TYPE	MASS
1	3732.04

***** ANSYS RESULTS INTERPRETATION (POST1) *****

ENTER /SHOW,DEVICE-NAME TO ENABLE GRAPHIC DISPLAY
ENTER FINISH TO LEAVE POST1

*** NOTE *** CP = 215.063 TIME= 10:00:18
An active coordinate system is not zero.
RSYS= 11 CSYS= 11 DSYS= 0.

USE LAST SUBSTEP ON RESULT FILE FOR LOAD CASE 0

SET COMMAND GOT LOAD STEP= 1 SUBSTEP= 1 CUMULATIVE ITERATION= 1
TIME/FREQUENCY= 1.0000
TITLE= 8-120B Cask Wall Model for Stiffness Verification

NSEL FOR LABEL= NODE FROM 813 TO 813 BY 1

1 NODES (OF 6747 DEFINED) SELECTED BY NSEL COMMAND.

PRINT DOF NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN COORDINATE SYSTEM 11

NODE	UX	UY	UZ
813	-0.60321E-02	-0.48761E-30	0.79634E-14

```

MAXIMUM ABSOLUTE VALUES
NODE          813          813          813
VALUE  -0.60321E-02-0.48761E-30 0.79634E-14

```

```
EXIT THE ANSYS POST1 DATABASE PROCESSOR
```

```
***** ROUTINE COMPLETED ***** CP =          215.718
```

```
***** ANSYS SOLUTION ROUTINE *****
```

c* Lead**

```

ESEL  FOR LABEL= REAL  FROM          2 TO          2 BY          1
      1728  ELEMENTS (OF          6840  DEFINED) SELECTED BY  ESEL  COMMAND.
ERSE  FOR LABEL= TYPE  FROM          1 TO          1 BY          1
      1728  ELEMENTS (OF          6840  DEFINED) SELECTED BY  ERSE  COMMAND.
SELECT      ALL NODES HAVING ANY ELEMENT IN ELEMENT SET.
      2500 NODES (OF          6747  DEFINED) SELECTED FROM
      1728 SELECTED ELEMENTS BY NELE COMMAND.
DELETE ALL SPECIFIED NODAL LOADS FROM NODE          1 TO          6747 BY          1
NUMBER OF NODAL LOADS DELETED=          0
SPECIFIED NODAL LOAD FX  FOR SELECTED NODES          2832 TO          2832 BY          1
REAL= -5000.00000      IMAG=  0.000000000
***** ANSYS SOLVE      COMMAND *****

```

S O L U T I O N O P T I O N S

```

PROBLEM DIMENSIONALITY. . . . . .3-D
DEGREES OF FREEDOM. . . . . UX  UY  UZ
ANALYSIS TYPE . . . . . .STATIC (STEADY-STATE)
NEWTON-RAPHSON OPTION . . . . . .PROGRAM CHOSEN
GLOBALLY ASSEMBLED MATRIX . . . . . .SYMMETRIC

```

L O A D S T E P O P T I O N S

```

LOAD STEP NUMBER. . . . . 1
TIME AT END OF THE LOAD STEP. . . . . 1.0000
NUMBER OF SUBSTEPS. . . . . 1
MAXIMUM NUMBER OF EQUILIBRIUM ITERATIONS. . . . 15
STEP CHANGE BOUNDARY CONDITIONS . . . . . NO
TERMINATE ANALYSIS IF NOT CONVERGED . . . . .YES (EXIT)
CONVERGENCE CONTROLS. . . . . .USE DEFAULTS
PRINT OUTPUT CONTROLS . . . . . .NO PRINTOUT
DATABASE OUTPUT CONTROLS. . . . . .ALL DATA WRITTEN

```

FOR THE LAST SUBSTEP

SOLUTION MONITORING INFO IS WRITTEN TO FILE= file.mntr

**** CENTER OF MASS, MASS, AND MASS MOMENTS OF INERTIA ****

CALCULATIONS ASSUME ELEMENT MASS AT ELEMENT CENTROID

TOTAL MASS = 11243.

CENTER OF MASS	MOM. OF INERTIA ABOUT ORIGIN	MOM. OF INERTIA ABOUT CENTER OF MASS
XC = 0.73502E-01	IXX = 0.2920E+08	IXX = 0.6906E+07
YC = 39.125	IYY = 0.1252E+08	IYY = 0.7436E+07
ZC = -21.258	IZZ = 0.2919E+08	IZZ = 0.1198E+08
	IXY = -0.3233E+05	IXY = 0.7458E-09
	IYZ = 0.9351E+07	IYZ = 0.4098E-07
	IZX = 0.1090E+05	IZX = -6668.

*** MASS SUMMARY BY ELEMENT TYPE ***

TYPE	MASS
1	11242.9

***** ANSYS RESULTS INTERPRETATION (POST1) *****

ENTER /SHOW,DEVICE-NAME TO ENABLE GRAPHIC DISPLAY
 ENTER FINISH TO LEAVE POST1

*** NOTE *** CP = 216.451 TIME= 10:00:19
 An active coordinate system is not zero.
 RSYS= 11 CSYS= 11 DSYS= 0.

USE LAST SUBSTEP ON RESULT FILE FOR LOAD CASE 0

SET COMMAND GOT LOAD STEP= 1 SUBSTEP= 1 CUMULATIVE ITERATION= 1
 TIME/FREQUENCY= 1.0000
 TITLE= 8-120B Cask Wall Model for Stiffness Verification

NSEL FOR LABEL= NODE FROM 2832 TO 2832 BY 1

1 NODES (OF 6747 DEFINED) SELECTED BY NSEL COMMAND.

PRINT DOF NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1 SUBSTEP= 1
 TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN COORDINATE SYSTEM 11

NODE	UX	UY	UZ
------	----	----	----

2832 -0.13943E-01 0.10943E-29-0.17871E-13

MAXIMUM ABSOLUTE VALUES

NODE 2832 2832 2832
VALUE -0.13943E-01 0.10943E-29-0.17871E-13

EXIT THE ANSYS POST1 DATABASE PROCESSOR

***** ROUTINE COMPLETED ***** CP = 217.091

c* Inner shell stiffness**

ESEL FOR LABEL= REAL FROM 3 TO 3 BY 1

1080 ELEMENTS (OF 6840 DEFINED) SELECTED BY ESEL COMMAND.

ERSE FOR LABEL= TYPE FROM 1 TO 1 BY 1

1080 ELEMENTS (OF 6840 DEFINED) SELECTED BY ERSE COMMAND.

SELECT ALL NODES HAVING ANY ELEMENT IN ELEMENT SET.

2294 NODES (OF 6747 DEFINED) SELECTED FROM
1080 SELECTED ELEMENTS BY NELE COMMAND.

DELETE ALL SPECIFIED NODAL LOADS FROM NODE 1 TO 6747 BY 1

NUMBER OF NODAL LOADS DELETED= 0

SPECIFIED NODAL LOAD FX FOR SELECTED NODES 5683 TO 5683 BY 1
REAL= -5000.00000 IMAG= 0.00000000

***** ANSYS SOLVE COMMAND *****

S O L U T I O N O P T I O N S

PROBLEM DIMENSIONALITY.3-D
DEGREES OF FREEDOM. UX UY UZ
ANALYSIS TYPESTATIC (STEADY-STATE)
NEWTON-RAPHSON OPTIONPROGRAM CHOSEN
GLOBALLY ASSEMBLED MATRIXSYMMETRIC

L O A D S T E P O P T I O N S

LOAD STEP NUMBER. 1
TIME AT END OF THE LOAD STEP. 1.0000
NUMBER OF SUBSTEPS. 1
MAXIMUM NUMBER OF EQUILIBRIUM ITERATIONS. 15
STEP CHANGE BOUNDARY CONDITIONS NO
TERMINATE ANALYSIS IF NOT CONVERGEDYES (EXIT)
CONVERGENCE CONTROLS.USE DEFAULTS
PRINT OUTPUT CONTROLSNO PRINTOUT
DATABASE OUTPUT CONTROLS.ALL DATA WRITTEN
FOR THE LAST SUBSTEP

SOLUTION MONITORING INFO IS WRITTEN TO FILE= file.mntr

**** CENTER OF MASS, MASS, AND MASS MOMENTS OF INERTIA ****

CALCULATIONS ASSUME ELEMENT MASS AT ELEMENT CENTROID

TOTAL MASS = 1634.3

CENTER OF MASS	MOM. OF INERTIA ABOUT ORIGIN	MOM. OF INERTIA ABOUT CENTER OF MASS
XC = 0.30170E-01	IXX = 0.4137E+07	IXX = 0.9844E+06
YC = 39.125	IYY = 0.1605E+07	IYY = 0.9537E+06
ZC = -19.959	IZZ = 0.4137E+07	IZZ = 0.1635E+07
	IXY = -1929.	IXY = -0.2797E-10
	IYZ = 0.1276E+07	IYZ = -0.7451E-08
	IZX = 608.4	IZX = -375.7

*** MASS SUMMARY BY ELEMENT TYPE ***

TYPE	MASS
1	1634.31

***** ANSYS RESULTS INTERPRETATION (POST1) *****

ENTER /SHOW,DEVICE-NAME TO ENABLE GRAPHIC DISPLAY
ENTER FINISH TO LEAVE POST1

*** NOTE *** CP = 217.731 TIME= 10:00:20
An active coordinate system is not zero.
RSYS= 11 CSYS= 11 DSYS= 0.

USE LAST SUBSTEP ON RESULT FILE FOR LOAD CASE 0

SET COMMAND GOT LOAD STEP= 1 SUBSTEP= 1 CUMULATIVE ITERATION= 1
TIME/FREQUENCY= 1.0000
TITLE= 8-120B Cask Wall Model for Stiffness Verification

NSEL FOR LABEL= NODE FROM 5683 TO 5683 BY 1

1 NODES (OF 6747 DEFINED) SELECTED BY NSEL COMMAND.

PRINT DOF NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN COORDINATE SYSTEM 11

NODE	UX	UY	UZ
5683	-0.24309E-01	0.67009E-31	-0.10943E-14

MAXIMUM ABSOLUTE VALUES

NODE	5683	5683	5683
VALUE	-0.24309E-01	0.67009E-31	-0.10943E-14

EXIT THE ANSYS POST1 DATABASE PROCESSOR

***** ROUTINE COMPLETED ***** CP = 218.308

c*** Combined stiffness

6840 ELEMENTS (OF 6840 DEFINED) SELECTED BY EALL COMMAND.

6747 NODES (OF 6747 DEFINED) SELECTED BY NALL COMMAND.

DELETE ALL SPECIFIED NODAL LOADS FROM NODE 1 TO 6747 BY 1

NUMBER OF NODAL LOADS DELETED= 3

SPECIFIED NODAL LOAD	FX	FOR SELECTED NODES	813 TO	813 BY	1
REAL=	-5000.00000	IMAG=	0.00000000		

***** ANSYS SOLVE COMMAND *****

S O L U T I O N O P T I O N S

PROBLEM DIMENSIONALITY. 3-D
 DEGREES OF FREEDOM. UX UY UZ
 ANALYSIS TYPESTATIC (STEADY-STATE)
 NEWTON-RAPHSON OPTIONPROGRAM CHOSEN
 GLOBALLY ASSEMBLED MATRIXSYMMETRIC

L O A D S T E P O P T I O N S

LOAD STEP NUMBER. 1
 TIME AT END OF THE LOAD STEP. 1.0000
 AUTOMATIC TIME STEPPING ON
 INITIAL NUMBER OF SUBSTEPS 1
 MAXIMUM NUMBER OF SUBSTEPS 1000
 MINIMUM NUMBER OF SUBSTEPS 1
 MAXIMUM NUMBER OF EQUILIBRIUM ITERATIONS. 15
 STEP CHANGE BOUNDARY CONDITIONS NO
 TERMINATE ANALYSIS IF NOT CONVERGED YES (EXIT)
 CONVERGENCE CONTROLS. USE DEFAULTS
 PRINT OUTPUT CONTROLS NO PRINTOUT
 DATABASE OUTPUT CONTROLS. ALL DATA WRITTEN
 FOR THE LAST SUBSTEP

**** CENTER OF MASS, MASS, AND MASS MOMENTS OF INERTIA ****

CALCULATIONS ASSUME ELEMENT MASS AT ELEMENT CENTROID

TOTAL MASS = 16609.

CENTER OF MASS	MOM. OF INERTIA ABOUT ORIGIN	MOM. OF INERTIA ABOUT CENTER OF MASS
XC = 0.63948E-01	IXX = 0.4334E+08	IXX = 0.1025E+08
YC = 39.125	IYY = 0.1890E+08	IYY = 0.1124E+08
ZC = -21.477	IZZ = 0.4333E+08	IZZ = 0.1791E+08
	IXY = -0.4156E+05	IXY = 0.2874E-07
	IYZ = 0.1396E+08	IYZ = 0.1546E-06
	IZX = 0.1414E+05	IZX = -8673.

*** MASS SUMMARY BY ELEMENT TYPE ***

TYPE	MASS
1	16609.2

***** ANSYS RESULTS INTERPRETATION (POST1) *****

ENTER /SHOW,DEVICE-NAME TO ENABLE GRAPHIC DISPLAY
ENTER FINISH TO LEAVE POST1

*** NOTE *** CP = 250.553 TIME= 10:00:38
An active coordinate system is not zero.
RSYS= 11 CSYS= 11 DSYS= 0.

USE LAST SUBSTEP ON RESULT FILE FOR LOAD CASE 0

SET COMMAND GOT LOAD STEP= 1 SUBSTEP= 1 CUMULATIVE ITERATION= 12
TIME/FREQUENCY= 1.0000
TITLE= 8-120B Cask Wall Model for Stiffness Verification

NSEL FOR LABEL= NODE FROM 813 TO 813 BY 1

1 NODES (OF 6747 DEFINED) SELECTED BY NSEL COMMAND.

PRINT DOF NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN COORDINATE SYSTEM 11

NODE	UX	UY	UZ
813	-0.35240E-02	-0.71334E-27	0.11650E-10

MAXIMUM ABSOLUTE VALUES

NODE	813	813	813
VALUE	-0.35240E-02	-0.71334E-27	0.11650E-10

APPENDIX 3

(1-CD)

Volume in drive E is ST-679
Volume Serial Number is 4AD3-9FAA

Directory of E:\

11/09/2011	05:42 PM	26,476,544	file.db
11/09/2011	10:00 AM	18,350,080	file.rst
11/09/2011	08:49 AM	131,992	file000.png
11/09/2011	08:51 AM	101,650	file001.png
11/09/2011	08:52 AM	179,159	file002.png
11/09/2011	10:00 AM	99,981	file003.png
11/09/2011	10:00 AM	106,097	file004.png
11/09/2011	10:00 AM	93,992	file005.png
11/09/2011	10:01 AM	94,843	file007.png
11/09/2011	10:04 AM	1,160,755	model.out
11/09/2011	10:04 AM	68,571	solution.out
	11 File(s)	46,863,664	bytes

Total Files Listed:

11 File(s)	46,863,664 bytes
0 Dir(s)	0 bytes free