1.0 OBJECTIVE

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

2.0 REFERENCES

1. Energy*Solutions* 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 = $2\times5{,}000/0.0060321 = 1.658\times10^6$ 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 = $2\times5{,}000/0.013943 = 717{,}206$ 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 = $2\times5{,}000/0.024309 = 411{,}370$ 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} = $2\times5{,}000/0.003524 = 2.838\times10^6$ 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:

$$k_1 + k_2 + k_3 = 1.658\times10^6 + 717{,}206 + 411{,}370 = 2.787\times10^6 \text{ lb/in}$$

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.

Figures

(7 Pages)

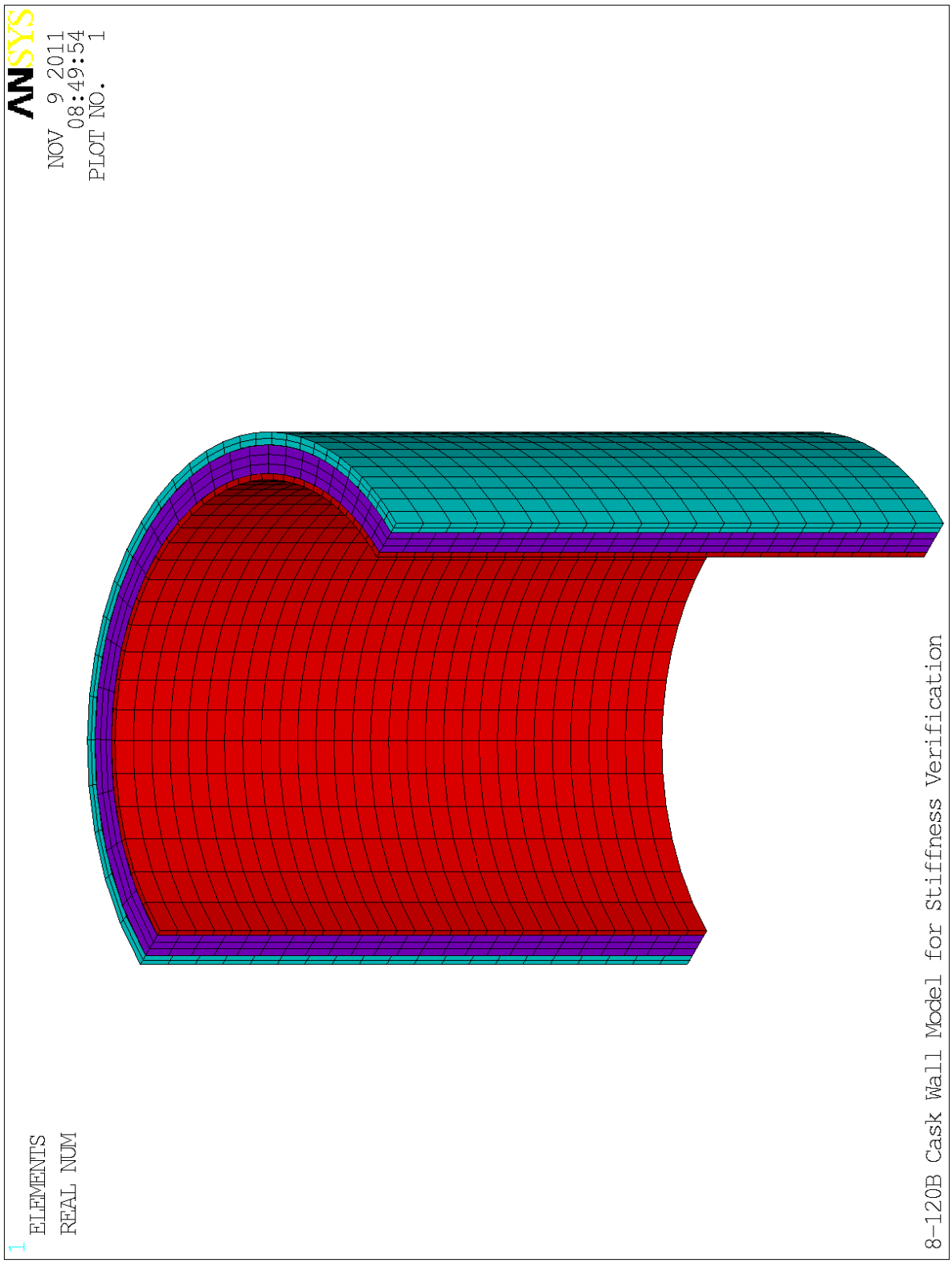

Figure 1 – 8-120B Cask Wall Finte Element Model

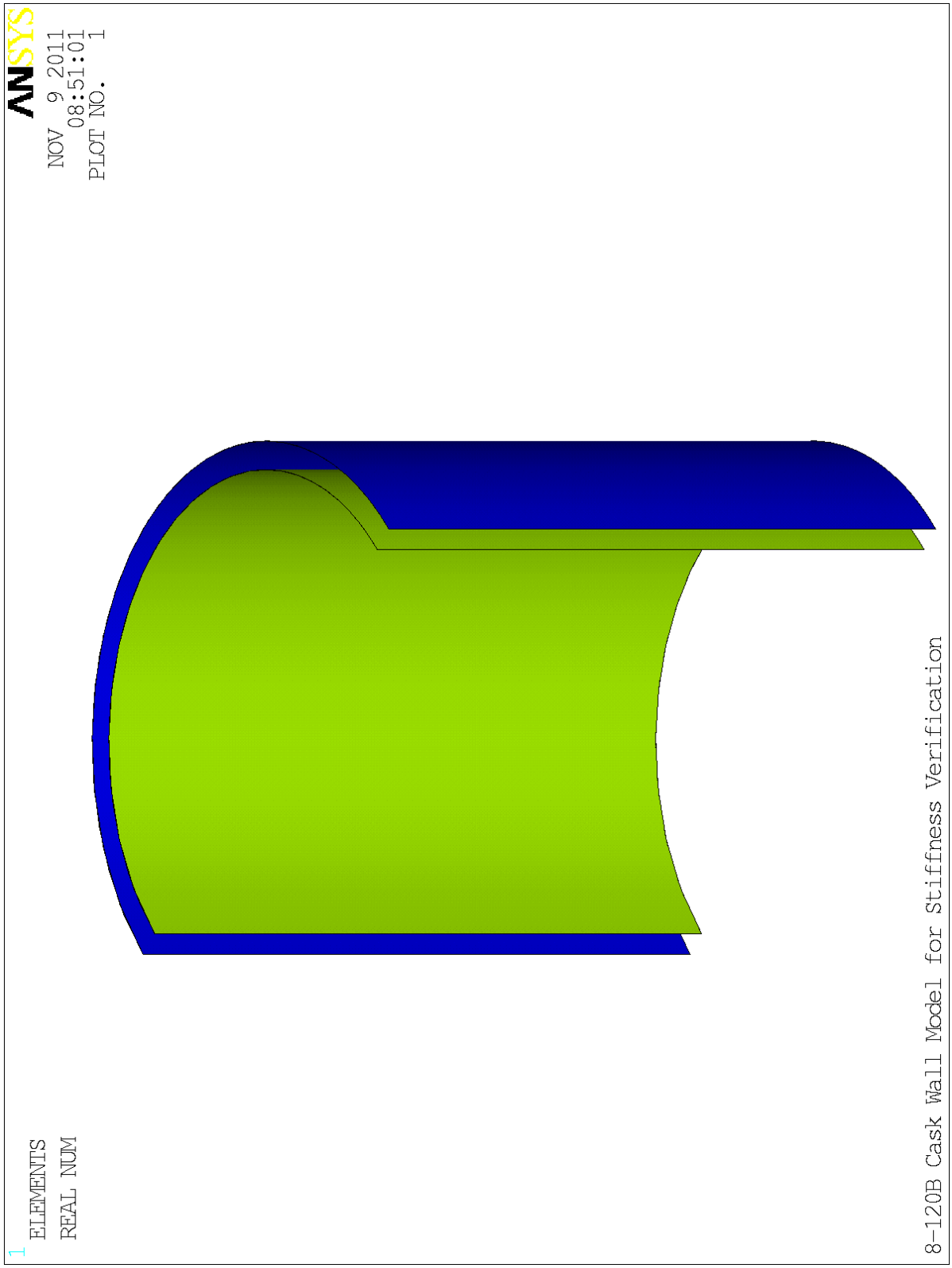

Figure 2 – Contact Elements Used in the Finte Element Model

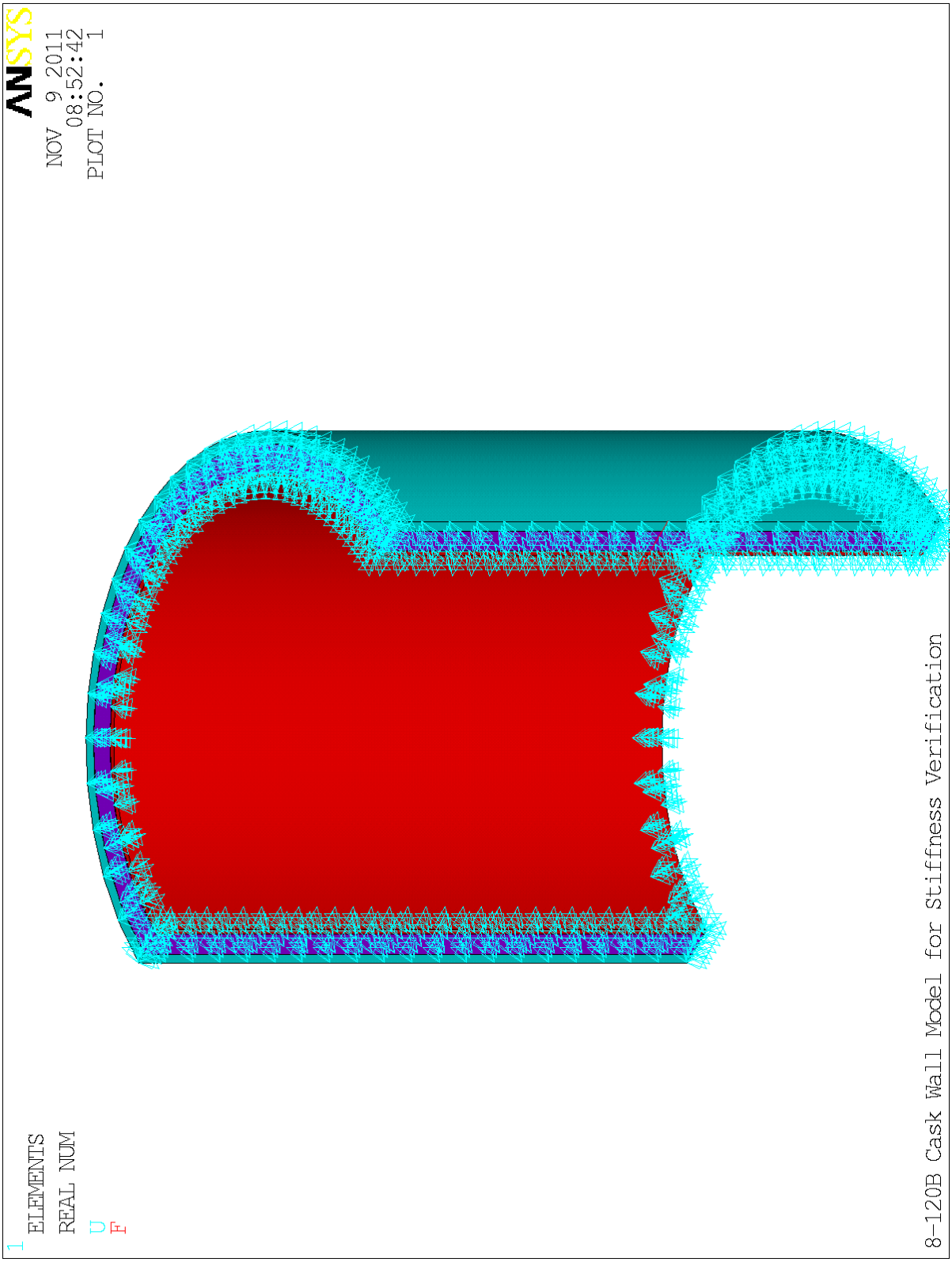

Figure 3 – Boundary Conditions Used in the Finte Element Model

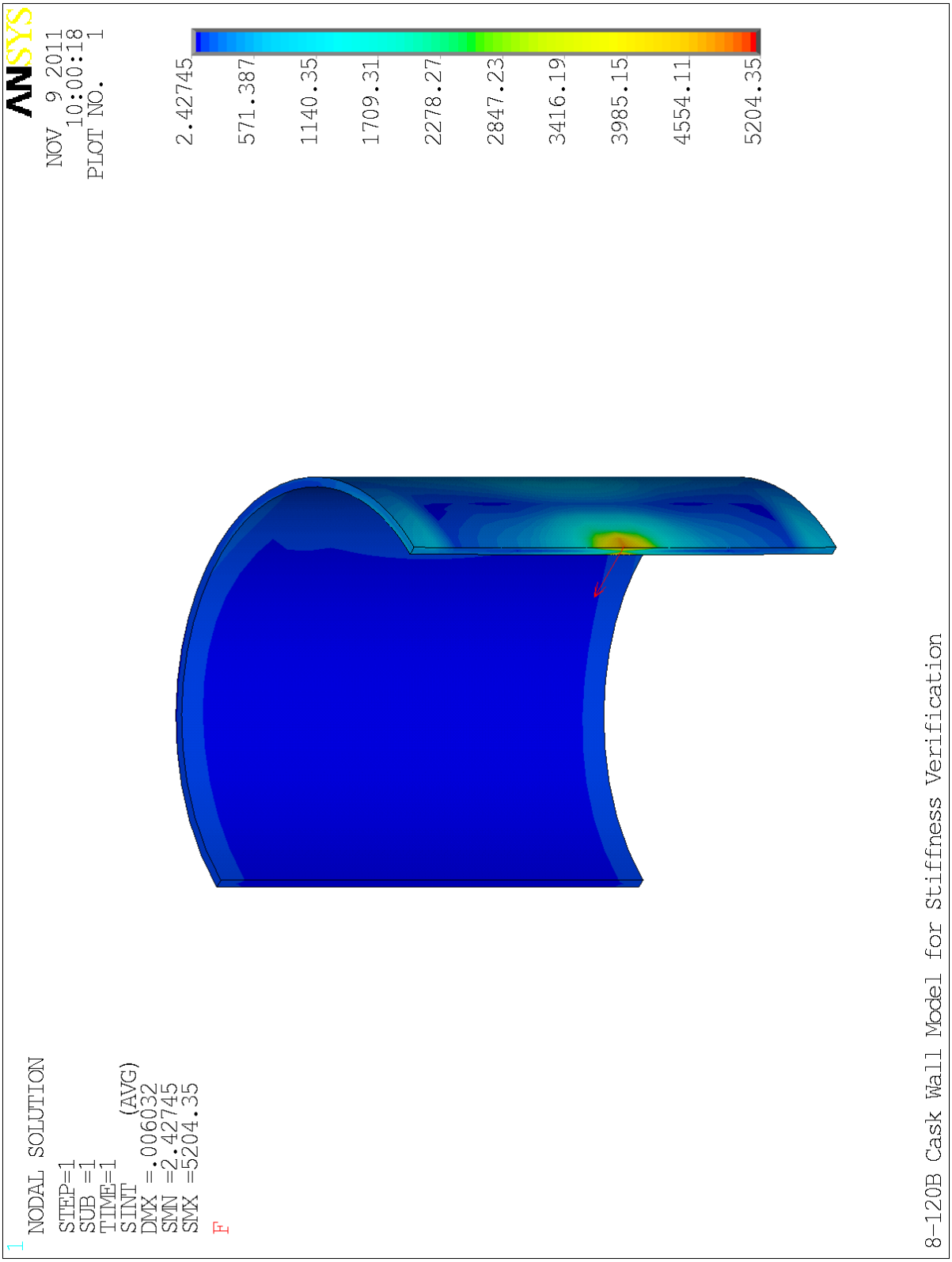

Figure 4 – Outer Shell under Applied Concentrated Load of 5,000 lb

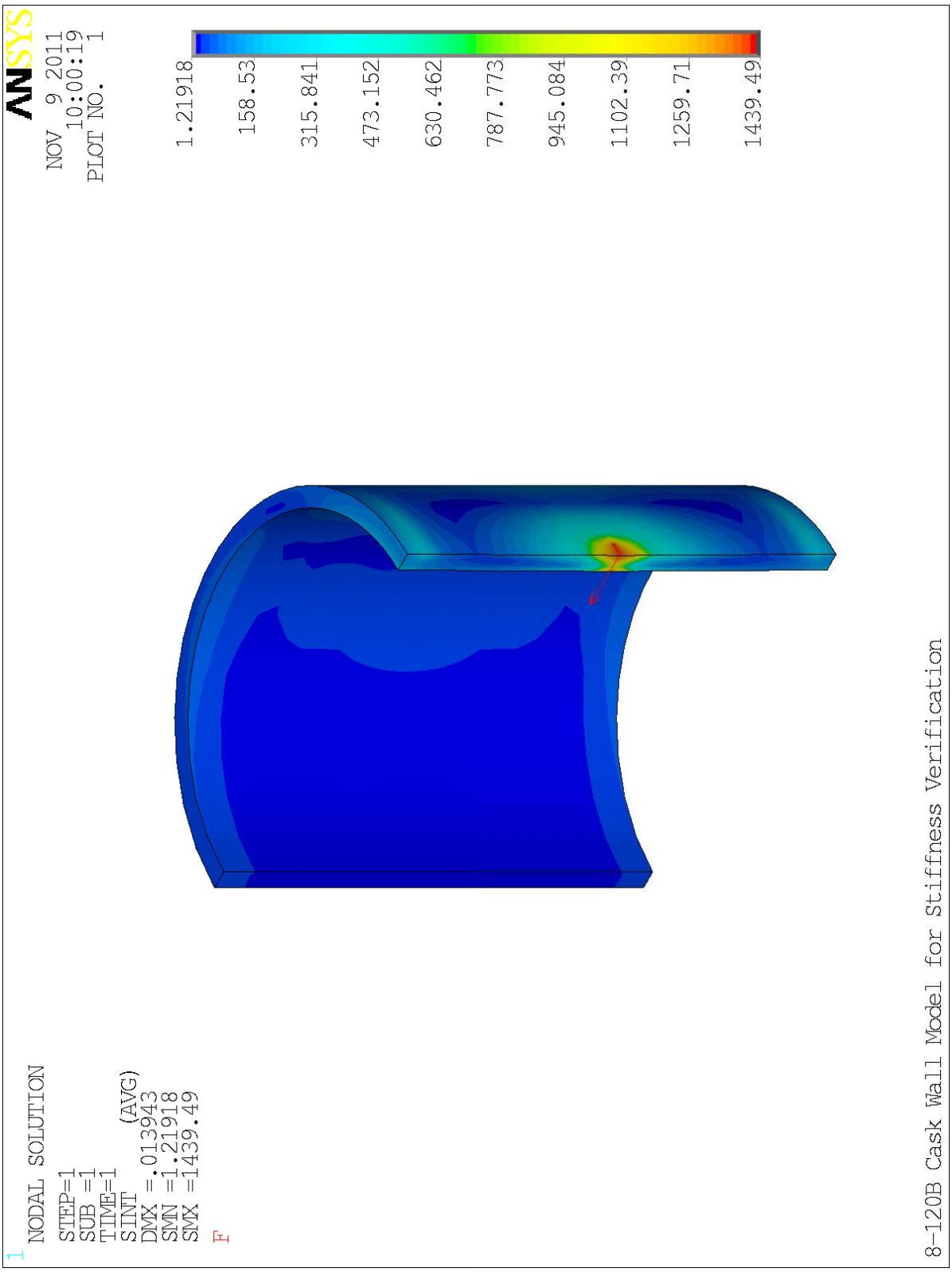

Figure 5 – Lead Shielding under Applied Concentrated Load of 5,000 lb

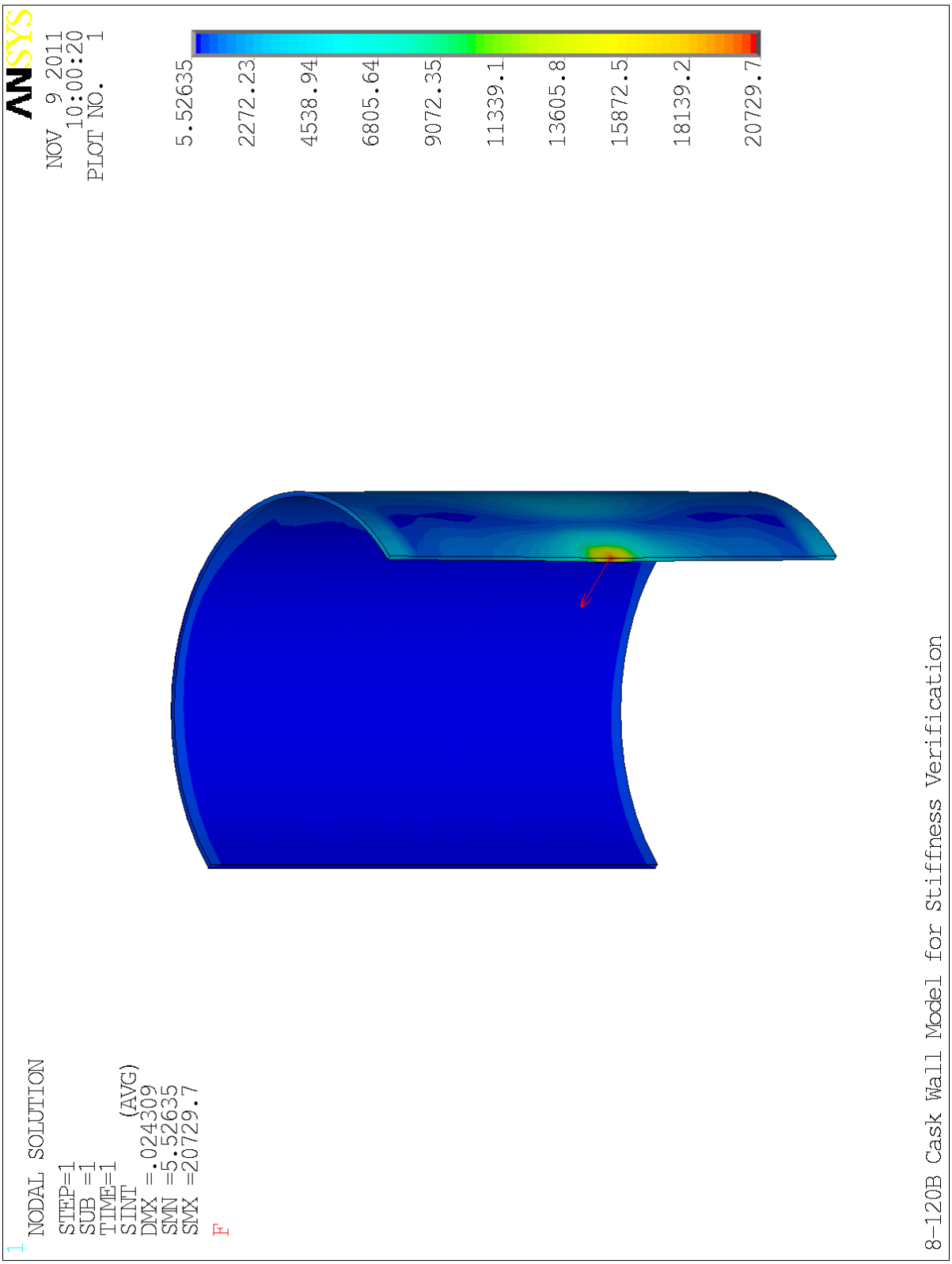

Figure 6 – Inner Shell under Applied Concentrated Load of 5,000 lb

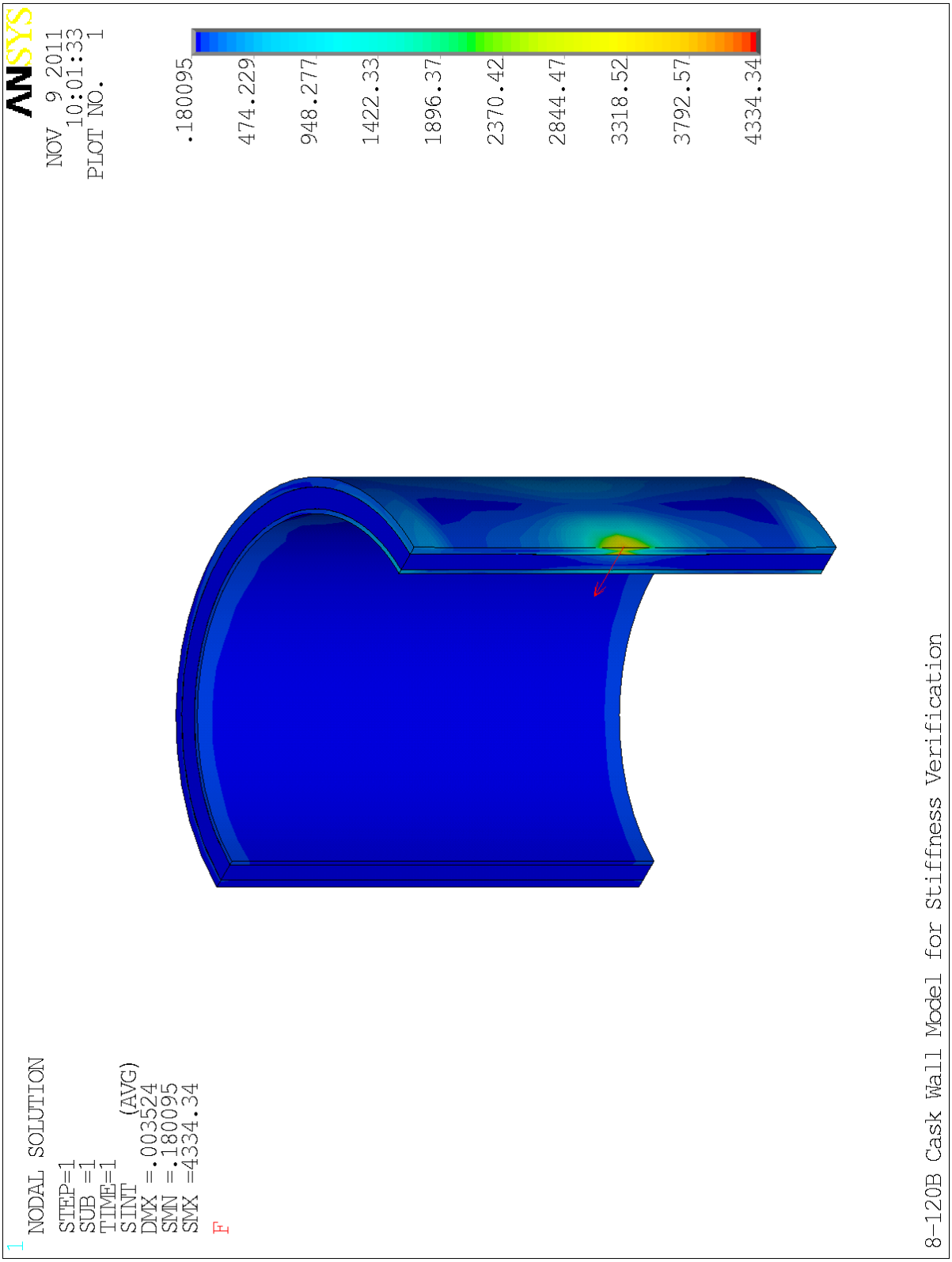

Figure 7 – 8-120B Composite Wall under Applied Concentrated Load of 5,000 lb

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 jobname . . . . . .file
 Initial jobname . . . . . .file

 Units . . . . . .unknown

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

 Finite element model summary:

                             Largest         Number        Number
                              Number        Defined      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.

B O U N D A R Y  C O N D I T I O N  I N F O R M A T I O N ----------------

                                      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

R O U T I N E  I N F O R M A T I O N ------------------------------------

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
```

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=  0.00000000

*****  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 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

                           MOM. OF INERTIA         MOM. OF INERTIA
 CENTER OF MASS             ABOUT ORIGIN        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=  0.00000000

*****  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 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.

                           MOM. OF INERTIA         MOM. OF INERTIA
 CENTER OF MASS             ABOUT ORIGIN        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 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 =   1634.3

                           MOM. OF INERTIA         MOM. OF INERTIA
 CENTER OF MASS             ABOUT ORIGIN        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 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
  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.

                          MOM. OF INERTIA         MOM. OF INERTIA
 CENTER OF MASS            ABOUT ORIGIN        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
```