PTC®

# Suggested Technique for Interpreting Pro/MECHANICA Structure ASCII Output Files

**Introduction**

This suggested technique describes the contents and format of the output files created by Pro/MECHANICA Structure. By default, these files are created in binary format so they are not directly readable. If the ASCII output file format is selected on the Run Settings form, the Pro/MECHANICA Structure engine will create the result files in ASCII format, which will allow the files to be viewed directly and utilized as input for other applications if desired.

**Procedure**

1. Begin by selecting the ASCII file format on the Run Settings Form as shown in Figure 1.

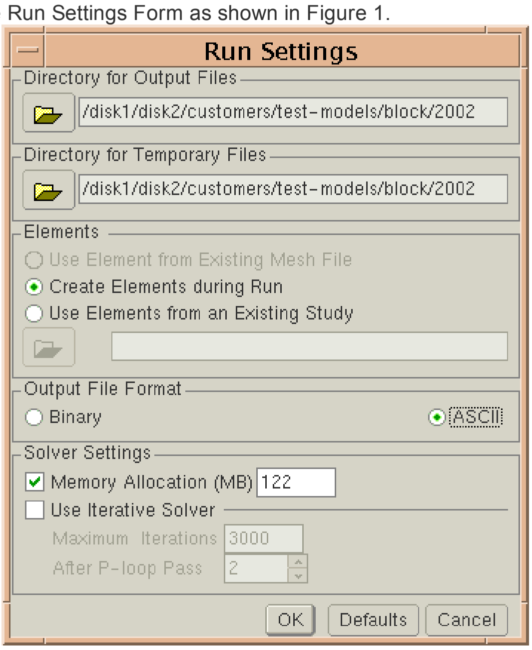


**Figure 1**

2. Once ASCII file format is selected, all the Pro/MECHANICA Structure engine output files will be created in ASCII file format. The content and formatting of these files is explained below:

(1) OUTPUT DIRECTORY TREE

The engine places output files in a directory called STUDY, where STUDY
is the name of the design study, and in subdirectories called ANLYS1,
ANLYS2, ..., where ANLYS1, ANLYS2, ... are the names of the analyses.
For dynamic time and frequency analyses more files may be placed in
ANLYS1/STEPn for each time or frequency step with full
postprocessing, where n corresponds to the master interval number
in the analysis definition. For large deformation static analyses,
files will be placed in ANLYS1/STEPn, where n corresponds to the load
interval number. For shock response, files will be placed
in ANLYS1/SHOCK. For transient thermal analysis, files will be placed
in ANLYS1/STEPn, where n corresponds to the master interval number.

A design study has one or more analyses. An analysis has one constraint
set and one or more load sets or modes.

A schematic representation of the output tree structure is shown below:

```
          STUDY
            |
           /^
          / |
         /  |
        /   |
       /    |
```

```
         /        |
        /         |
       /          |
      /           |
study.err      ANLYS1
study.rpt        | ------------------------------ SHOCK
study.stt        | ----------- STEPn              |
study.pnu      study.neu         |                |
study.dia      study.d##         |              study.d01
study.pas      study.s##      study.d##         study.s01
study.hst      study.r##      study.s##         study.p01
               study.a##      study.p##         study.a01
               study.f##      study.a##         study.n01
               study.t##      study.v##         study.b01
               study.cnv      study.w##
               study.res      study.x##
               study.l##      study.y##
               study.g##      study.n##
               study.opt      study.h##
               study.ter      study.i##
               study.coe      study.j##
               study.tld      study.k##
               study.p##      study.m##
               study.n##      study.q##
               study.c##      study.cnv
               study.mor      study.tld
               study.buc      study.b##
               study.b##
               study.fatigue##
               study.ss##
```

The list of files shown above is the list of all possible output files.
Some of these files may not be created depending on the analysis options,
analysis type and design study type.

(2) DISPLACEMENT/STRESS, TEMPERATURE/FLUX POST-PROCESSING FILES

A uniform grid is created and laid on top of the geometric element model for the purpose of post-processing. This grid splits up the geometric elements into smaller regions of the same kind: quadrilateral geometric elements are split up into quadrilateral regions, brick geometric elements are split up into brick regions, e.t.c.. The only exception are tetrahedral geometric elements that are split up into tetrahedral and octahedral regions.

In this document the geometric elements are referred to as "p-elements" while the regions defined by the grid are referred to as "h-elements". The nodes that are part of the geometric element model are referred to as "p-nodes" while the nodes of the grid are referred to as "h-nodes". Note that the "h-nodes" that are also "p-nodes" are numbered consistently in both sets.

```
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/study.pnu
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"p-nodes"      pnod
"p-elements"     pnel
   iel  iej  nod1  nod2  nod3  nod4  nod5  nod6  nod7  nod8
   iel  iej  nod1  nod2  nod3  nod4  nod5  nod6  nod7  nod8
   iel  iej  nod1  nod2  nod3  nod4  nod5  nod6  nod7  nod8
    "
```

```
     "...
```

Notes: Connectivity of the geometric element model

```
     pnod: total number of p-nodes
     pnel: total number of p-elements
     iel: p-element number
     iej: total number of edges of this p-element;
         e.g. for a quadrilateral element iej=4
     nod1-nod8: the numbers of the nodes defining this p-element;
              n/a node numbers are set equal to zero;
              e.g. for a quadrilateral element nod5...nod8=0
```

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

study/analysis/study.neu

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"h-nodes"        hnod
     inod   x   y   z
      iind  inod1  inod2 i nod3  inod4  inod5  inod6  inod7  inod8
     inod   x   y   z
      iind  inod1  inod2  inod3  inod4  inod5  inod6  inod7  inod8
     inod   x   y   z
      iind  inod1  inod2  inod3  inod4  inod5  inod6  inod7  inod8
       "
       "...
"h-elements"     hnel
    iel  iej  nod1  nod2  nod3  nod4  nod5  nod6  nod7  nod8
    iel  iej  nod1  nod2  nod3  nod4  nod5  nod6  nod7  nod8
    iel  iej  nod1  nod2  nod3  nod4  nod5  nod6  nod7  nod8
     "
     "...
```

Notes: Connectivity of the grid

```
     hnod: total number of h-nodes
     inod: h-node number
     x,y,z: coordinates of this h-node in global rectangular system
     iind: indicator categorizing this h-node as follows:
      iind=0: this h-node is a p-node
      iind=1: this h-node is internal to a p-element edge
      iind=2: this h-node is internal to a p-element tri. face
      iind=3: this h-node is internal to a p-element quad. face
      iind=4: this h-node is internal to a tetrahedron p-element
      iind=5: this h-node is internal to a wedge p-element
      iind=6: this h-node is internal to a brick p-element
     inod1-inod8: p-node numbers defining the p-elements, p-
               element faces and p-element edges referred to by
               the indicator iind;
               n/a node numbers are set equal to zero;
     hnel: total number of h-elements
     iel: h-element number
     iej: total number of edges of this h-element;
         e.g. for a quadrilateral element iej=4;
         octahedral h-elements have iej=-12 so that they can be
         distinguished from bricks that also have 12 edges
     nod1-nod8: the numbers of the nodes defining this h-element;
              n/a node numbers are set equal to zero;
              e.g. for a quadrilateral element nod5...nod8=0
```

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

study/analysis/study.mor

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"material_orientations"
      iel   inod
      e1_x e1_y e1_z e2_x e2_y e2_z

      iel   inod
      e1_x e1_y e1_z e2_x e2_y e2_z
      .
      .
      .
```

Or in column notation this is:

```
iel   inod
      mo_01 mo_02 mo_03 mo_04 mo_05 mo_06
```

Notes: this is the material orientation file

All quantities are calculated at the h-node locations.

All quantities are reported with respect to
the WCS.

Note that h-nodes that are common to more than one
p-element will be assigned more than one value set (one
for each p-element).

Only h-nodes that belong to elements with material orientations
(3d solids, 3d shells, 2d solids, 2d plates)

iel: p-element number
inod: h-node number

mo_01-03 e1_x,y,z : WCS components of the first material orientation basis unit vector
mo_04-05 e2_x,y,z : WCS components of the second material orientation basis unit vector

The third material orientation basis unit vector is found from e3 = e1 X e2

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.d##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

IN STRUCTURAL ANALYSES

```
"displacements"   iset   nset   nrbm   dmax   f    name
     inod   dx   dy   dz
     inod   dx   dy   dz
     inod   dx   dy   dz
       "
       "...
```

Notes: Displacements in static, modal, dynamic time, dynamic frequency
shock and buckling analyses. For static, modal, and buckling
analysis the file is
placed in ANLYS#. For dynamic time and frequency analysis the
file is placed in STEP####. For shock analysis it is placed in
SHOCK.

##: load set for static dynamic time and dynamic frequency
mode number for modal and buckling (two digit format)
always 01 for shock
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
nrbm: number of rigid body modes
dmax: maximum magnitude of displacement in the model

f: frequency of this mode if modal analysis,
buckling load factor if buckling analysis,
frequency of calculation if dynamic frequency response
time of calculation if dynamic time response
0 if static or other dynamic analyses
name: load set name (not for modal, buckling or shock)
inod: h-node number
dx,dy,dz: displacements of this h-node in global rectangular
system

IN THERMAL ANALYSES

```
"temperatures"   iset   nset   tmax   time   name
     inod   t
     inod   t
     inod   t
       "
       "...
```

Notes: Temperatures. For steady-state thermal analysis, the file is
placed in ANLYS#. For transient thermal analysis the file is
placed in STEP####.

##: load set in two digit format
iset: load set number; equal to ##
nset: total number of load sets
tmax: maximum temperature in the model
time: time of master interval if transient thermal analysis
0 if steady-state thermal analysis
name: load set name
inod: h-node number
t: temperature

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.a##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"rotations"   iset   nset   thmax   f    name
     inod   thx   thy   thz
     inod   thx   thy   thz
     inod   thx   thy   thz
       "
       "...
```

Notes: Rotations in static, modal, dynamic time, dynamic frequency analysis,
shock, and buckling analyses. For static, modal, and buckling
analysis the file is
placed in ANLYS#. For dynamic time and frequency analysis the
file is placed in STEP####. For shock analysis it is placed in
SHOCK.

##: load set for static dynamic time and dynamic frequency
mode number for modal and buckling (two digit format)
always 01 for shock
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
thmax: maximum magnitude of rotation in the model
f: frequency of this mode if modal analysis,
buckling load factor if buckling analysis,
frequency of calculation if dynamic frequency response
time of calculation if dynamic time response
0 if static or other dynamic analyses
name: load set name (not for modal, buckling or shock)
inod: h-node number
thx,thy,thz: rotations of this h-node in global rectangular
system

```
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.s##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

IN STRUCTURAL ANALYSES

"stresses"   iset   nset   name
      iel   inod   ind nvals
         s1    s2    s3    s4    s5    s6
         s7    s8    s9    s10   s11   s12
         s13   s14   s15   s16   s17   s18
         s19   s20   s21   s22   s23   s24
         s25   s26   s27   s28   s29   s30
         s31   s32   s33   s34   s35   s36
         s37   s38   s39   s40   s41   s42
         s43   s44   s44   s45   s46   s47
         s48   s49   s50   s51   s52   s53
      iel   inod   ind nvals
         s1    s2    s3    s4    s5    s6
         s7    s8    s9    s10   s11   s12
         s13   s14   s15   s16   s17   s18
         s19   s20   s21   s22   s23   s24
         s25   s26   s27   s28   s29   s30
         s31   s32   s33   s34   s35   s36
         s37   s38   s39   s40   s41   s42
         s43   s44   s44   s45   s46   s47
         s48   s49   s50   s51   s52   s53
      iel   inod   ind nvals
         s1    s2    s3    s4    s5    s6
         s7    s8    s9    s10   s11   s12
         s13   s14   s15   s16   s17   s18
         s19   s20   s21   s22   s23   s24
         s31   s32   s33   s34   s35   s36
         s37   s38   s39   s40   s41   s42
         s43   s44   s44   s45   s46   s47
         s48   s49   s50   s51   s52   s53
        "
        "...
```

Notes: Stress/strain distribution in static,modal, dynamic time,
dynamic frequency, shock, or buckling analysis.
All stresses and strains
are calculated at the h-node locations and are reported with
respect to the global rectangular coordinate system with the
following exception, shell membrane and bending stresses are
reported with respect to the local material coordinate system.
Forces and moments for beams are also reported with respect to the
local (defined by the p-element's orientation) coordinate system.
Note that h-nodes that are common to more than one p-element will
be assigned more than one stress/strain value set (one for each
p-element). Also note that top and bottom surfaces of plate
p-elements are defined by their connectivity using the
right-hand-rule.

For static, modal, and buckling analysis file is placed in
ANLYS#. For dynamic time and frequency the file is placed in
STEP####. For shock analysis it is placed in SHOCK.

##: load set for static dynamic time and dynamic frequency
mode number for modal and buckling (two digit format)
always 01 for shock
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
name: load set name (not for modal or shock)
iel: p-element number

inod: h-node number
ind: =1 if 3-D beams; =2 if 3-D or 2-D shells
=3 if 3-D solids or 2-D solids or plates
nvals : number of values to follow for this record (can vary by
element type if desired) minimum == 38, max == 53
s1: global (strain)xx for solids and 2-D surface elements;
global (strain)xx on the top surface for shells and
line 2-D elements;
global (force)x for beams
s2: global (strain)yy for solids and 2-D surface elements;
global (strain)yy on the top surface for shells and
line 2-D elements;
global (force)y for beams
s3: global (strain)xy for solids and 2-D surface elements;
global (strain)xy on the top surface for shells and
line 2-D elements;
global (force)z for beams
s4: global (strain)zz for solids and 2-D surface elements;
global (strain)zz on the top surface for shells and
line 2-D elements;
global (moment)x for beams
s5: global (strain)yz for solids and 2-D surface elements;
global (strain)yz on the top surface for shells and
line 2-D elements;
global (moment)y for beams
s6: global (strain)xz for solids and 2-D surface elements;
global (strain)xz on the top surface for shells and
line 2-D elements;
global (moment)z for beams
s7: zero for solids and 2-D surface elements;
global (strain)xx on the bottom surface for shells and
line 2-D elements;
local (force)x for beams
s8: zero for solids and 2-D surface elements;
global (strain)yy on the bottom surface for shells and
line 2-D elements;
local (force)y for beams
s9: zero for solids and 2-D surface elements;
global (strain)xy on the bottom surface for shells and
line 2-D elements;
local (force)z for beams
s10: zero for solids and 2-D surface elements;
global (strain)zz on the bottom surface for shells and
line 2-D elements;
local (moment)x for beams
s11: zero for solids and 2-D surface elements;
global (strain)yz on the bottom surface for shells and
line 2-D elements;
local (moment)y for beams
s12: zero for solids and 2-D surface elements;
global (strain)xz on the bottom surface for shells and
line 2-D elements;
local (moment)z for beams
s13: global (stress)xx for solids and 2-D surface elements;
global (stress)xx on the top surface for shells and
line 2-D elements;
axial stress at (-1,-1) or point 1 cross-sectional point for beams
s14: global (stress)yy for solids and 2-D surface elements;
global (stress)yy on the top surface for shells and
line 2-D elements;
axial stress at (0,-1) or point 2 cross-sectional point for beams
s15: global (stress)xy for solids and 2-D surface elements;
global (stress)xy on the top surface for shells and
line 2-D elements;
axial stress at (+1,-1) or point 3 cross-sectional point for beams
s16: global (stress)zz for solids and 2-D surface elements;
global (stress)zz on the top surface for shells and

line 2-D elements;
axial stress at (-1,0) or point 4 cross-sectional point for beams
s17: global (stress)yz for solids and 2-D surface elements;
global (stress)yz on the top surface for shells and
line 2-D elements;
axial stress at (0,0) or point 5 cross-sectional point for beams
s18: global (stress)xz for solids and 2-D surface elements;
global (stress)xz on the top surface for shells and
line 2-D elements;
axial stress at (+1,0) or point 6 cross-sectional point for beams
s19: zero for solids and 2-D surface elements;
global (stress)xx on the bottom surface for shells and
line 2-D elements;
axial stress at (-1,+1) or point 7 cross-sectional point for beams
s20: zero for solids and 2-D surface elements;
global (stress)yy on the bottom surface for shells and
line 2-D elements;
axial stress at (0,+1) or point 8 cross-sectional point for beams
s21: zero for solids and 2-D surface elements;
global (stress)xy on the bottom surface for shells and
line 2-D elements;
axial stress at (+1,+1) or point 9 cross-sectional point for beams
s22: zero for solids and 2-D surface elements;
global (stress)zz on the bottom surface for shells and
line 2-D elements;
tensile stress for beams
s23: zero for solids and 2-D surface elements;
global (stress)yz on the bottom surface for shells and
line 2-D elements;
bending stress (most +ve in cross-section) for beams
s24: zero for solids and 2-D surface elements;
global (stress)xz on the bottom surface for shells and
line 2-D elements;
axial force (most +ve in cross-section) for beams
s25: zero for solids and 2-D surface elements;
Von Mises stress on the top surface for shells and
line 2-D elements;
axial force (most -ve in cross-section) for beams
This field contains contact pressure for contact
analyses only.
s26: zero for solids and 2-D surface elements;
Von Mises stress on the bottom surface for shells and
line 2-D elements;
torsional shear stress for beams
s27: Von Mises stress for solids and 2-D surface elements;
max. Von Mises stress for shells and line 2-D elements;
von Mises stress (max over cross-section) for beams
s28: zero for solids and 2-D surface elements;
max. Principal stress on the top surface for shells and
line 2-D elements;
bending stress (y) for beams
s29: zero for solids and 2-D surface elements;
max. Principal stress on the bottom surface for shells and
line 2-D elements;
bending stress (z) for beams
s30: max. Principal stress for solids and 2-D surface elements;
max. Principal stress for shells and line 2-D elements;
max. Principal stress (max over cross-section) for beams
s31: zero for solids and 2-D surface elements;
membrane strain energy/unit area for shells and
line 2-D elements;
tensile strain energy per unit length for beams
s32: zero for solids and 2-D surface elements;
bending strain energy/unit area for shells and
line 2-D elements;
bending strain energy per unit length for beams
s33: zero for solids and 2-D surface elements;

shear strain energy/unit area for shells and
line 2-D elements;
shear strain energy per unit length for beams
s34: zero for solids and 2-D surface elements;
membrane/bending strain energy for shells;
zero for line 2-D elements;
torsional strain energy per unit length for beams
s35: Strain Energy/unit volume for solids and 2-D
surface elements;
total strain energy/unit area for shells and
line 2-D elements;
total strain energy per unit length for beams
s36: zero for solids and 2-D surface elements;
minimum principal stress (top) for shells and
line 2D elements;
tensile strain for beams
s37: zero for solids and 2-D surface elements;
minimum principal stress (bottom) for shells and
line 2D elements;
torsional strain for beams
s38: minimum principal stress for solids and 2-D surface elements;
minimum principal stress ( minimum of top and bottom)
for shells and line 2D elements;
min. Principal stress (min over cross-section) for beams
s39: zero for solids and 2-D surface elements;
local midsurface stress (xz) for shells
bending strain (y) for beams
s40: zero for solids and 2-D surface elements;
local midsurface stress (yz) for shells
bending strain (z) for beams
s41: zero for solids and 2-D surface elements;
membrane stress (xx) for shells
zero for beams
s42: zero for solids and 2-D surface elements;
membrane stress (yy) for shells
zero for beams
s43: zero for solids and 2-D surface elements;
membrane stress (xy) for shells
zero for beams
s44: zero for solids and 2-D surface elements;
bending stress (xx) for top surface of shells
zero for beams
s45: zero for solids and 2-D surface elements;
bending stress (yy) for top surface of shells
zero for beams
s46: zero for solids and 2-D surface elements;
bending stress (xy) for top surface of shells
zero for beams
s47: zero for solids and 2-D surface elements;
bending stress (xx) for bottom surface of shells
zero for beams
s48: zero for solids and 2-D surface elements;
bending stress (yy) for bottom surface of shells
zero for beams
s49: zero for solids and 2-D surface elements;
bending stress (xy) for bottom surface of shells
zero for beams
s50: zero for solids and 2-D surface elements;
transverse shear shear (x) for top surface of shells
zero for beams
s51: zero for solids and 2-D surface elements;
transverse shear stress (y) for top surface of shells
zero for beams
s52: zero for solids and 2-D surface elements;
transverse shear shear (x) for bottom surface of shells
zero for beams
s53: zero for solids and 2-D surface elements;

```
        transverse shear stress (y) for bottom surface of shells
        zero for beams
```

IN THERMAL ANALYSES

```
"fluxes"   iset   nset   name
      iel   inod
        s1   s2   s3   s4   s5   s6
      iel   inod
        s1   s2   s3   s4   s5   s6
      iel   inod
        s1   s2   s3   s4   s5   s6
       "
       "...
```

Notes: temperature gradient/heat flux distribution. All gradients and fluxes are calculated at the h-node locations and are reported with respect to the global rectangular coordinate system. Note that h-nodes that are common to more than one p-element will be assigned more than one gradient/flux value set (one for each p-element).

For steady-state thermal analysis the file is placed in ANLYS#. For transient thermal analysis the file is placed in STEP####.

##: load set in two digit format
iset: load set; equal to ##
nset: total number of load sets
name: load set name
iel: p-element number
inod: h-node number
s1: dT/dx
s2: dT/dy
s3: dT/dz
s4: (heat flux)x
s5: (heat flux)y
s6: (heat flux)z

```
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.p##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
"principal_vects"
      iel   inod
        s1   ex   ey   ez
        s2   ex   ey   ez
        s3   ex   ey   ez
        s4   ex   ey   ez
      iel   inod
        s1   ex   ey   ez
        s2   ex   ey   ez
        s3   ex   ey   ez
        s4   ex   ey   ez
      iel   inod
        s1   ex   ey   ez
        s2   ex   ey   ez
        s3   ex   ey   ez
        s4   ex   ey   ez
       "
       "...
```

IN THERMAL ANALYSES

```
"fluxes"   iset   nset   name
      iel   inod
         s1   s2   s3   s4   s5   s6
      iel   inod
         s1   s2   s3   s4   s5   s6
      iel   inod
         s1   s2   s3   s4   s5   s6
        "
        "...
```

Notes: temperature gradient/heat flux distribution. All gradients
and fluxes are calculated at the h-node locations and
are reported with respect to the global rectangular coordinate
system. Note that h-nodes that are common to more than
one p-element will be assigned more than one gradient/flux
value set (one for each p-element).

For steady-state thermal analysis the file is placed in
ANLYS#. For transient thermal analysis the file is placed in
STEP####.

##: load set in two digit format
iset: load set; equal to ##
nset: total number of load sets
name: load set name
iel: p-element number
inod: h-node number
s1: dT/dx
s2: dT/dy
s3: dT/dz
s4: (heat flux)x
s5: (heat flux)y
s6: (heat flux)z

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

study/analysis/study.p##

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"principal_vects"
      iel   inod
         s1   ex   ey   ez
         s2   ex   ey   ez
         s3   ex   ey   ez
         s4   ex   ey   ez
      iel   inod
         s1   ex   ey   ez
         s2   ex   ey   ez
         s3   ex   ey   ez
         s4   ex   ey   ez
      iel   inod
         s1   ex   ey   ez
         s2   ex   ey   ez
         s3   ex   ey   ez
         s4   ex   ey   ez
       "
       "...
```

Notes: Maximum/minimum principal stress directions in static,
modal, dynamic time, dynamic frequency, shock or buckling analysis.
All principal stresses are calculated at the h-node locations
and their directions are reported with respect to the global
rectangular coordinate system. Note that h-nodes
that are common to more than one p-element will be
assigned more than one principal stress value set (one
for each p-element). Only h-nodes that belong to quad
or tri elements are included.

```
    For static, modal and buckling analysis file is placed
    in ANLYS#.  For dynamic time and frequency the file is
    placed in STEP####.  For shock analysis it is placed in SHOCK.

    ##: load set for static dynamic time and dynamic frequency
        mode number for modal and buckling (two digit format)
        always 01 for shock
    iset: load set or mode number; equal to ##
    nset: total number of load sets or modes
    name: load set name (not for modal or shock)
    iel: p-element number
    inod: h-node number
    s1: max principal stress on the top surface for 3-D shells;
        max principal stress for 2-D surface elements
    s2: min principal stress on the top surface for 3-D shells;
        min principal stress for 2-D surface elements
    s3: max principal stress on the bottom surface for 3-D shells
    s4: min principal stress on the bottom surface for 3-D shells
    ex, ey, ez: unit vector w.r.t. global cartesian coordinates




::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.n##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"Shell_Results"  iset  name
    iel   inod
    g_xx          g_xy         g_yy
    g_max_prin_val g_max_prin_x  g_max_prin_y  g_max_prin_z
    g_min_prin_val g_min_prin_x  g_min_prin_y  g_min_prin_z
    k_xx          k_xy         k_yy
    k_max_prin_val k_max_prin_x  k_max_prin_y  k_max_prin_z
    k_min_prin_val k_min_prin_x  k_min_prin_y  k_min_prin_z
    o_x           o_y
    N_xx          N_xy         N_yy
    N_max_prin_val N_max_prin_x  N_max_prin_y  N_max_prin_z
    N_min_prin_val N_min_prin_x  N_min_prin_y  N_min_prin_z
    M_xx          M_xy         M_yy
    M_max_prin_val M_max_prin_x  M_max_prin_y  M_max_prin_z
    M_min_prin_val M_min_prin_x  M_min_prin_y  M_min_prin_z
    Q_x           Q_y

    iel   inod
    g_xx                g_xy        g_yy
    g_max_prin_val g_max_prin_x  g_max_prin_y  g_max_prin_z
    g_min_prin_val g_min_prin_x  g_min_prin_y  g_min_prin_z
    .
    .
    .

Or in column notation this is:

iel   inod
    sr_01        sr_02        sr_03
    sr_04        sr_05        sr_06    sr_07
    sr_08        sr_09        sr_10    sr_11
    sr_12        sr_13        sr_14
    sr_15        sr_16        sr_17    sr_18
    sr_19        sr_20        sr_21    sr_22
    sr_23        sr_24
    sr_25        sr_26        sr_27
    sr_28        sr_29        sr_30    sr_31
    sr_32        sr_33        sr_34    sr_35
    sr_36        sr_37        sr_38
    sr_39        sr_40        sr_41    sr_42
```

```
    sr_43      sr_44      sr_45    sr_46
    sr_47      sr_48
```

Notes: Shell results in static, modal, dynamic time, dynamic
frequency, shock or buckling analysis.

All quantities are calculated at the h-node locations.

All tensor quantities except the principal direction vectors are reported with respect to the material orientation basis of the element.
The principal direction vectors are reported with respect to the WCS.

Note that h-nodes that are common to more than one p-element will be assigned more than one value set (one for each p-element).

Only h-nodes that belong to 3d shells are included.

For static, modal and buckling analysis file is placed in ANLYS#. For dynamic time and frequency the file is placed in STEP####. For shock analysis it is placed in SHOCK.

```
##: load set for static dynamic time and dynamic frequency
   mode number for modal and buckling (two digit format)
   always 01 for shock

iel: p-element number
inod: h-node number


sr_01-03    g_xx,xy,yy     : membrane (midsurface) strain
sr_04      g_max_prin_val  : max principal membrane strain value
sr_05-07    g_max_prin_x,y,z: max principal membrane strain vector
sr_08      g_min_prin_val  : min principal membrane strain value
sr_09-11    g_min_prin_x,y,z: min principal membrane strain vector
sr_12-14    k_xx, k_xy, k_yy: curvature change
sr_15      k_max_prin_val  : max principal curvature change value
sr_16-18    k_max_prin_x,y,z: max principal curvature change vector
sr_19      k_min_prin_val  : min principal curvature change value
sr_20-22    k_min_prin_x,y,z: min principal curvature change vector

sr_23-24    o_x,y          : transverse shear strain

sr_25-27    N_xx,xy,yy     : membrane resultant force
sr_28      N_max_prin_val  : max principal membrane resultant force value
sr_29-31    N_max_prin_x,y,z: max principal membrane resultant force vector
sr_32      N_min_prin_val  : min principal membrane resultant force value
sr_33-35    N_min_prin_x,y,z: min principal membrane resultant force vector
sr_36-38    M_xx, M_xy, M_yy: resultant moment
sr_39      M_max_prin_val  : max principal resultant moment value
sr_40-42    M_max_prin_x,y,z: max principal resultant moment vector
sr_43      M_min_prin_val  : min principal resultant moment value
sr_44-46    M_min_prin_x,y,z: min principal resultant moment vector
sr_47-48    Q_x,y          : transverse shear force
```

```
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.b##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"ply_stresses"  iset  is_complex  maj_vers  revision  name
    iel       inod    n_plies
  ply_num   orientation
```

```
 s_xx_top_Re  s_yy_top_Re  s_zz_top_Re  s_xy_top_Re  s_xz_top_Re  s_yz_top_Re
 s_xx_bot_Re  s_yy_bot_Re  s_zz_bot_Re  s_xy_bot_Re  s_xz_bot_Re  s_yz_bot_Re
 e_xx_top_Re  e_yy_top_Re  g_zz_top_Re  e_xy_top_Re  g_xz_top_Re  g_yz_top_Re
 e_xx_bot_Re  e_yy_bot_Re  g_zz_bot_Re  e_xy_bot_Re  g_xz_bot_Re  g_yz_bot_Re
 s_xx_top_Im  s_yy_top_Im  s_zz_top_Im  s_xy_top_Im  s_xz_top_Im  s_yz_top_Im
 s_xx_bot_Im  s_yy_bot_Im  s_zz_bot_Im  s_xy_bot_Im  s_xz_bot_Im  s_yz_bot_Im
 e_xx_top_Im  e_yy_top_Im  g_zz_top_Im  e_xy_top_Im  g_xz_top_Im  g_yz_top_Im
 e_xx_bot_Im  e_yy_bot_Im  g_zz_bot_Im  e_xy_bot_Im  g_xz_bot_Im  g_yz_bot_Im
"
"...
```

Notes: Laminate stress/strain distribution in static, modal, dynamic time,
dynamic frequency, dynamic random, dynamic shock or buckling analysis.

All quantities are calculated at the h-node locations and
reported with respect to the global rectangular coordinate system.

For static, modal, and buckling analysis file is placed in ANLYS#.
For dynamic time and frequency the file is placed in STEP####.
For shock analysis it is placed in SHOCK.
For dynamic random the file is placed in RMS.

##: load set for static dynamic time and dynamic frequency
mode number for modal and buckling (two digit format)
always 01 for shock

iset: load set or mode number; equal to ##.
is_complex: 1 for dynamic frequency and random analyses,
0 for all other analyses.
maj_vers: Pro/Mechanica version #.
revision: revision # in maj_vers.
name: load set name (not for modal or shock)
iel: p-element number
inod: h-node number
n_plies: number of plies for element iel.
ply_num: ply number.
orientation: orientation of ply with respect to it's
material 3 direction.

s_(xx,yy,xy,zz,yz,xz)_top_Re: Real components of stress tensor at
top of the lamina.
s_(xx,yy,xy,zz,yz,xz)_bot_Re: Real components of stress tensor at
bottom of the lamina.
s_(xx,yy,xy,zz,yz,xz)_top_Im: Imag. components of stress tensor at
top of the lamina.
Output only if is_complex is 1.
s_(xx,yy,xy,zz,yz,xz)_bop_Im: Imag. components of stress tensor at
bottom of the lamina.
Output only if is_complex is 1.
e_xx,e_yy,g_xy,e_zz,g_yz,g_xz_top_Re:
Real components of strain tensor at
top of the lamina.
e_xx,e_yy,g_xy,e_zz,g_yz,g_xz_bot_Re:
Real components of strain tensor at
bottom of the lamina.
e_xx,e_yy,g_xy,e_zz,g_yz,g_xz_top_Im:
Real components of strain tensor at
top of the lamina.
Output only if is_complex is 1.
e_xx,e_yy,g_xy,e_zz,g_yz,g_xz_bop_Im:
Real components of strain tensor at
bottom of the lamina.
Output only if is_complex is 1.

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.h##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"displacements"  iset   nset   dmax   f    name
     inod   dx   dy   dz
     inod   dx   dy   dz
     inod   dx   dy   dz
       "
       "...
```

Notes: Phases of displacement in dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
dmax: 0
f: frequency of calculation
name: load set name
inod: h-node number
dx,dy,dz: phases of displacement of this h-node in global rectangular system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.v##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"velocities"   iset   nset   vmax   f    name
     inod   vx   vy   vz
     inod   vx   vy   vz
     inod   vx   vy   vz
       "
       "...
```

Notes: Velocities in dynamic time or dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
vmax: maximum magnitude of velocity in the model
f: frequency of calculation if dynamic frequency response
time of calculation if dynamic time response
name: load set name
inod: h-node number
vx,vy,vz: velocities of this h-node in global rectangular system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.i##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"velocities"   iset   nset   vmax   f    name
     inod   vx   vy   vz
     inod   vx   vy   vz
     inod   vx   vy   vz
       "
       "...
```

Notes: Phases of velocity in dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes

vmax: 0
f: frequency of calculation
name: load set name
inod: h-node number
vx,vy,vz: phases of velocity of this h-node in global rectangular
system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.w##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"accelerations"   iset   nset   amax   f    name
     inod   ax   ay   az
     inod   ax   ay   az
     inod   ax   ay   az
       "
       "...
```

Notes: Accelerations in dynamic time or dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
amax: maximum magnitude of acceleration in the model
f: frequency of calculation if dynamic frequency response
time of calculation if dynamic time response
name: load set name
inod: h-node number
ax,ay,az: accelerations of this h-node in global rectangular
system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.j##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"accelerations"   iset   nset   wmax   f    name
     inod   wx   wy   wz
     inod   wx   wy   wz
     inod   wx   wy   wz
       "
       "...
```

Notes: Phases of acceleration in dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
wmax: 0
f: frequency of calculation
name: load set name
inod: h-node number
wx,wy,wz: phases of acceleration of this h-node in global rectangular
system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.k##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"rotations"   iset   nset   amax   f    name
```

```
    inod   ax   ay   az
    inod   ax   ay   az
    inod   ax   ay   az
     "
     "...
```

Notes: Phases of rotation in dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
amax: 0
f: frequency of calculation
name: load set name
inod: h-node number
ax,ay,az: phases of rotation of this h-node in global rectangular system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.x##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"rotat vel"   iset   nset   vmax   f    name
    inod   vx   vy   vz
    inod   vx   vy   vz
    inod   vx   vy   vz
     "
     "...
```

Notes: Rotational velocities in dynamic time or dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
vmax: maximum magnitude of rotational velocity in the model
f: frequency of calculation if dynamic frequency response
time of calculation if dynamic time response
name: load set name
inod: h-node number
vx,vy,vz: rotational velocities of this h-node in global rectangular system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.m##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"rotat vel"   iset   nset   vmax   f    name
    inod   vx   vy   vz
    inod   vx   vy   vz
    inod   vx   vy   vz
     "
     "...
```

Notes: Phases of rotational velocity in dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
vmax: 0
f: frequency of calculation

name: load set name
inod: h-node number
vx,vy,vz: phases of rotational velocity of this h-node in
global rectangular system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.y##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"rotat accel"   iset   nset   amax   f    name
     inod   ax   ay   az
     inod   ax   ay   az
     inod   ax   ay   az
       "
       "...
```

Notes: Rotational accelerations in dynamic time or dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
amax: maximum magnitude of rotational acceleration in the model
f: frequency of calculation if dynamic frequency response
time of calculation if dynamic time response
name: load set name
inod: h-node number
ax,ay,az: rotational accelerations of this h-node in global rectangular
system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.q##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"rotat accel"   iset   nset   wmax   f    name
     inod   wx   wy   wz
     inod   wx   wy   wz
     inod   wx   wy   wz
       "
       "...
```

Notes: Phases of rotational acceleration in dynamic frequency analysis.
The file is placed in STEP####.

##: load set
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
wmax: 0
f: frequency of calculation
name: load set name
inod: h-node number
wx,wy,wz: phases of rotational acceleration of this h-node in
global rectangular system

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.r##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"analysis type" antyp
"reactions"   iset   nset    name
"resultant"   rx     ry     rz
```

```
"nodes"      nnodr
     inod   rx     ry     rz     mx     my     mz
       "
       "...
"edges"      nedgr  nplot "no_curmpc" (or "yes_curmpc")
     nod1   nod2
     nod1  rx     ry     rz     mx     my     mz
     nod2  rx     ry     rz     mx     my     mz
     nod#  rx     ry     rz     mx     my     mz
       "
       "...
```

Notes: Reactions in static, buckling or modal analysis

antyp: analysis type
iset: load set or mode number; equal to ##
nset: total number of load sets or modes
name: load set name (if static analysis only)
rx,ry,rz,mx,my,mz: real values of reactions at a point
nnodr: number of nodes which have reactions
inod: h-node number
nedgr: number of edges which have reactions
nplot: number of plotting points per edge
"yes_cmpc": mpc's were created because of constraints
in curvilinear coordinates
"no_cmpc": no mpc's due to curvilinear coordinates
nod1, nod2: p-node numbers of edge
nod#: h-node numbers on interior of edge

(3) HISTORY FILE

```
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/study.hst
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

msg Updating design variables
update_parms  npar  iflag
    idv1          par
    idv2          par
     "
     "
    npar          iflag
    idv1          par
    idv2          par
     "
    ...
```

Notes: Parameter values for major model updates during an
optimization or sensitivity design study.  Steps for line
searches or derivative calculations are not included.

npar: number of updated parameters; equals the number of lines
for each update
iflag = 1 if final update
= 0 if not final update
idv1, idv2 ...: parameter dbid
par: value of parameter

(4) X-Y PLOTTING FILES

```
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
```

```
study/analysis/study.res
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"Measure Convergence Plotting File"

"Analysis:"   anname

ncol   "columns"
nset   "rows"

"col"        "quantity"
 1           "p-loop pass number"
 2           measname     measdbid
    "
    "

"DATA"
ip   v1    v2    v3    v4    v5
    v6    v7    v8    v9    ...
   "
   "
ip   v1    v2    v3    v4    v5
    v6    v7    v8    v9    ...
   "
   "
ip   v1    v2    v3    v4    v5
    v6    v7    v8    v9    ...
   "
   "


Notes: Values of measures at each iteration of the p-loop for all
      load sets or modes

      anname: analysis name
      ncol: total number of columns
      nset: number of loads sets or modes; equals the number of sets
           of values at each p-level
      measdbid: dbid of the measure
      measname: name of measure
      ip: p-loop iteration
      v1, v2, v3...: values of measures


:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.f##
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"frequency response"

"Analysis:"   anname

ncol   "columns"
nset   "rows"

"col"        "quantity"
 1           "frequency value"
 2           measname     measdbid
    "
    "

"DATA"
fre   v1    v2    v3    v4    v5
    v6    v7    v8    v9    ...
   "
   "
fre   v1    v2    v3    v4    v5
```

```
    v6    v7    v8    v9    ...
  "
  "
fre   v1    v2    v3    v4    v5
    v6    v7    v8    v9    ...
  "
  "
```

Notes: Values of measures at each frequency value of a frequency
response

anname: analysis name
ncol: total number of columns
nset: =1
measdbid: dbid of the measure
measname: name of measure
fre: frequency value
v1, v2, v3...: values of measures

```
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.t##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"time response"

"Analysis:"   anname

ncol   "columns"
nset   "rows"

"col"       "quantity"
 1          "time value"
 2          measname     measdbid
   "
   "

"DATA"
tim   v1    v2    v3    v4    v5
    v6    v7    v8    v9    ...
  "
  "
tim   v1    v2    v3    v4    v5
    v6    v7    v8    v9    ...
  "
  "
tim   v1    v2    v3    v4    v5
    v6    v7    v8    v9    ...
  "
  "
```

Notes: Values of measures at each time value of a time
response

anname: analysis name
ncol: total number of columns
nset: =1
measdbid: dbid of the measure
measname: name of measure
tim: time value
v1, v2, v3...: values of measures

```
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.g##
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"Global Sensitivity Plotting File"

"Parameter:"  pname    pdbid

ncol   "columns"
nset   "rows"
nstep  "steps

"col"       "quantity"
 1          "Parameter: pname"
 2          measname     measdbid
   "
   "

"DATA"
pval   v1    v2    v3    v4     v5
      v6    v7    v8    ...
   "

pval   v1    v2    v3    v4     v5
      v6    v7    v8    ...
   "

pval   v1    v2    v3    v4     v5
      v6    v7    v8    ...
   "

   "...

Notes: Plotting file for global sensitivity; values of measures
      at each parameter step.

      ##: parameter number in two digit format
      pname: parameter name
      pdbid: parameter dbid
      ncol: total number of columns
      nset: number of loads sets or modes; equals the number of
           sets of values at each parameter step
      nstep: number of parameter steps
      measdbid: dbid of the measure
      measname: name of measure
      pval: parameter value
      v1, v2, v3...: values of measures


:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.l##
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"Local Sensitivity Plotting File"

"Parameter:"  pname    pdbid

ncol   "columns"
nset   "rows"
nstep  "steps

"col"       "quantity"
 1          "Parameter: pname"
 2          measname     measdbid
   "
   "
```

```
"DATA"
pval   v1    v2    v3   v4    v5
     v6    v7    v8    ...
  "

pval   v1    v2    v3   v4    v5
     v6    v7    v8   ...
  "
```

Notes: Plotting file for local sensitivity; values of measures at
the two ends of the parameter range.

##: parameter number in two digit format
pname: parameter name
pdbid: parameter dbid
ncol: total number of columns
nset: number of loads sets or modes; equals the number of
sets of values at each parameter step
nstep: number of parameter steps; nstep=2
measdbid: dbid of the measure
measname: name of measure
pval: parameter value
v1, v2, v3...: values of measures

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.opt
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"Optimization Plotting File"

ncol   "columns"
nset   "rows"

"col"        "quantity"
 1           "optimization iteration number"
 2           measname     measdbid
   "
   "

"DATA"
iter   v1    v2    v3    v4    v5
     v6    v7    v8    ...
  "
  "
iter   v1    v2    v3    v4    v5
     v6    v7    v8    ...
  "
  "
```

Notes: Plotting file for optimization; values of measures at
every step of the optimization loop.

ncol: total number of columns
nset: number of loads sets or modes; equals the number of
sets of values at each parameter step
measdbid: dbid of the measure
measname: name of measure
iter: optimization loop iteration number
v1, v2, v3...: values of measures

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.c##

```
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"Contact Plotting File"

ncol   "columns"
nloadinc  "load increments"

"col"       "quantity"
 1          "Load increment"
 2          measname     measdbid
   "
   "

"DATA"
loadinc   v1    v2    v3    v4     v5
      v6    v7    v8    ...
   "

Notes: Plotting file for contact values of measures
      at each load increment

      ##: load set number in two digit format
      ncol: total number of columns
      nloadinc: number of load increments
      measdbid: dbid of the measure
      measname: name of measure
      loadinc: load increment value (floating point number)
      v1, v2, v3...: values of measures


(5) DIAGNOSTIC FILES


:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/study.err
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

Notes: Input data echo and fatal errors encountered during run time.


:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.ter
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

Notes: Input data echo and fatal errors encountered during run time.
      This file is produced for thermal analyses only.


:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/study.rpt
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

Notes: Human readable file which contains a log of the progress of
      analyses or optimization design studies, numerical values
      of measures, warning messages, or error messages.


:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/study.stt
:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

Notes: Human readable file which contains the start and completion
      times of major steps of the engine run.


:::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/study.pas
```

```
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

Notes: Human readable file which contains the start and completion
       times of major steps in the engine run. (in more detailed form
       than study/study.stt)


::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/study.dia
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

Notes: File for communicating an error code to the post-processor in
       the event of a fatal error during the engine run.


::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.cnv
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

"Applied Structure Version 3.0(00)"          IF STRUCTURAL ANALYSIS
"Applied Thermal Version 1.0(00)"            IF THERMAL ANALYSIS
"Convergence Report"
date/time stamp

"Analysis:" anname

nel     "elements"
nedge   "edges"

"Convergence History:"
"  * number of load cases"           IF STATIC OR THERMAL ANALYSIS
"  * number of modes"                IF MODAL (DYNAMIC) OR BUCKLING ANALYSIS
"  * total strain energy"            IF STATIC ANALYSIS
"  * frequency"                      IF MODAL (DYNAMIC) ANALYSIS
"  * buckling load factor"           IF BUCKLING ANALYSIS
"  * total gradient energy"          IF THERMAL ANALYSIS
"  * errors in energy norms"
"  * max error in energy norm"
"  * max local temp & energy error"
"  * convergence index"
"  * total number of equations"
"  * number of changed elements"
"  * max p-order of any edge"
"  * p-order of edges"
"  * clock time"

"p-loop start time:"
date/time stamp

"---- p-loop pass:  1 ----"
int
long
long
long
long
long
int
int
int
  int    int    int    int    int    int    int    int
  int    int    int    int    int    int    int    int
  ...

date/time stamp

"---- p-loop pass:  2 ----"
...
```

```
...

"---- p-loop pass:  3 ----"
...
...
"
"

"The analysis (did not) converged to" icon "on"
convergence_criterion

IF STATIC ANALYSIS

"Final convergence results, displacements:"
"  edge  node 1  node 2   p-order   dU/Umax    U/Umax   l.c."
   int   int     int      int       long       long     int   d/r (*)
  "
  "

IF MODAL (DYNAMIC) OR BUCKLING  ANALYSIS

"Final convergence results, displacements:"
"  edge  node 1  node 2   p-order   dU/Umax    U/Umax   mode"
   int   int     int      int       long       long     int   d/r (*)
  "
  "

IF THERMAL ANALYSIS

"Final convergence results, temperatures:"
"  edge  node 1  node 2   p-order   dT/Tmax    T/Tmax   l.c."
   int   int     int      int       long       long     int   d (*)
  "
  "

IF STATIC OR THERMAL ANALYSIS

"Final convergence results, element energy:"
" element   edges        sqrt(dE/E)      E/Etot     l.c."
    int      int          long           long       int (*)
  "
  "

 IF MODAL (DYNAMIC) OR BUCKLING ANALYSIS

"Final convergence results, element energy:"
" element   edges        sqrt(dE/E)      E/Etot     mode"
    int      int          long           long       int (*)
  "
  "
```

Notes: This file contains convergence information at each iteration
of the p-loop, including:
the p-order of each edge
errors in edge displacements or temperatures
strain energies or frequencies or gradient energies
the convergence index

At the end it reports and edges and elements for which convergence was not achieved.

For transient thermal analysis, the .cnv file is placed in the STEP#### directory. It contains only the p-order of each edge at the time of the master interval.

(6) SCRATCH FILES

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.tld
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

Notes: File for passing thermal loads to structural analyses. The file
is created only for thermal analyses.

For transient thermal analysis, the .tld file is placed in the
STEP#### directory. It contains the thermal field at the time
of the master interval.

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.coe
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

Notes: File for storing the function coefficients of the solution. The
file is used by dynamic analyses referring to previously run
model or dynamic analyses.

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.buc
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

Notes: Written by any static analyses for use in a subsequent
buckling analysis.
Contains static analysis solution info needed to reconstruct
element stress during buckling analysis element stress-stiffness
matrix computation.

::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.fatigue##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

```
"fatigues"   iset   nset   name
      iel   inod   ind
         s1    s2    s3    s4    s5    s6
         s7    s8    s9    s10
```

Notes: Results for a fatigue analysis.
All quantities are calculated at the h-node locations lying on
the external surface of the model and are reported with respect
to the global rectangular coordinate system.

iset: load set number; equal to ##
nset: total number of load sets or modes
name: load set name
iel: p-element number
inod: h-node number
ind: =2 if 3-D shells, =3 if 3-D solids

s1: Log of life for solids
  Log of life on the top surface for shells
s2: Log of damage for solids
  Log of damage on the top surface for shells
s3: Factor of Safety for solids
  Factor of Safety on the top surface for shells
s4: Biaxiality Ratio for solids
  Biaxiality Ratio on the top surface for shells
s5: Confidence for solids
  Confidence on the top surface for shells
s6: 0 for solids
  Log of life on the bottom surface for shells

```
    s7: 0 for solids
       Log of damage on the bottom surface for shells
    s8: 0 for solids
       Factor of Safety on the bottom surface for shells
    s9: 0 for solids
       Biaxiality Ratio on the bottom surface for shells
    s10: 0 for solids
        Confidence on the bottom surface for shells


::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::
study/analysis/study.ss##
::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::::

IN STRUCTURAL ANALYSES

"stresses"   iset   nset   name
    iel   inod   ind nvals
      s1    s2    s3    s4    s5    s6
      s7    s8    s9    s10   s11   s12
      s13   s14   s15   s16   s17   s18
      s19   s20   s21   s22   s23   s24
      s25   s26   s27   s28   s29   s30
      s31   s32   s33   s34   s35   s36
      s37   s38   s39   s40   s41   s42
      s43   s44   s44   s45   s46   s47
      s48   s49   s50   s51   s52   s53
    iel   inod   ind nvals
      s1    s2    s3    s4    s5    s6
      s7    s8    s9    s10   s11   s12
      s13   s14   s15   s16   s17   s18
      s19   s20   s21   s22   s23   s24
      s25   s26   s27   s28   s29   s30
      s31   s32   s33   s34   s35   s36
      s37   s38   s39   s40   s41   s42
      s43   s44   s44   s45   s46   s47
      s48   s49   s50   s51   s52   s53

Notes:
    Stress components distribution in a fatigue analysis in ASCII.

    This is output only if the environment variable WRITE_SURFACE_STRESSES is set.

    The format is the same as that described for the .s## file, except that only
    the following slots are filled, the rest are all 0.

    s13: global (stress)xx for solids;
       global (stress)xx on the top surface for shells
    s14: global (stress)yy for solids;
       global (stress)yy on the top surface for shells
    s15: global (stress)xy for solids;
       global (stress)xy on the top surface for shells
    s16: global (stress)zz for 3d solids ;
       global (stress)zz on the top surface for shells
    s17: global (stress)yz for solids;
       global (stress)yz on the top surface for shells
    s18: global (stress)xz for 3d solids
       global (stress)xz on the top surface for shells
    s19: zero for solids;
       global (stress)xx on the bottom surface for shells
    s20: zero for solids;
       global (stress)yy on the bottom surface for shells
    s21: zero for solids;
       global (stress)xy on the bottom surface for shells
    s22: zero for solids;
       global (stress)zz on the bottom surface for shells
    s23: zero for solids;
```

global (stress)yz on the bottom surface for shells
s24: zero for solids;
global (stress)xz on the bottom surface for shells