/CWD

/CWD, *DIRPATH*
Changes the current working directory.

SESSION: Run Controls
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

DIRPATH
The full path name of the new working directory.

Notes

After issuing the /CWD command, all new files opened with no default directory specified (via the **FILE**, **/COPY**, or **RESUME** commands, for example) default to the new *DIRPATH* directory.

If issuing the command interactively and the specified directory does not exist, no change of directory occurs and the command generates an error message. If the command executes via a batch run and the specified directory does not exist, the batch program terminates with an error.

Menu Paths

Utility Menu>File>Change Directory

/FILNAME

/FILNAME, *Fname*, *Key*
Changes the Jobname for the analysis.

SESSION: Run Controls
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Fname

Name (32 characters maximum) to be used as the **Jobname**. Defaults to the initial **Jobname** as specified on the ANSYS execution command, or to **file** if none specified.

Key

Specify whether to use the existing log, error, lock, and page files or start new files.

0, OFF — Continue using current log, error, lock, and page files.

1, ON — Start new log, error, lock, and page files (old log and error files are closed and saved, but old lock and page files are deleted). Existing log and error files are appended.

Notes

All subsequently created files will be named with this **Jobname** if *Key* = 0. Use *Key* = 1 to start new log, error, lock, and page files. The previous **Jobname** is typically defined on the ANSYS program execution line (see the *Operations Guide*). This command is useful when different groups of files created throughout the run are to have different names. For example, the command may be used

before each substructure pass to avoid overwriting files or having to rename each file individually.

This command is valid only at the Begin level.

Menu Paths

Utility Menu>File>Change Jobname

/TITLE

/TITLE, *Title*
Defines a main title.

DATABASE: Set Up
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Title

Input up to 72 alphanumeric characters. Parameter substitution may be forced within the title by enclosing the parameter name or parametric expression within percent (%) signs.

Notes

The title is carried through the printout and written on various files. The title written to a file is the title defined at that time. Special characters may be used within the title text. Subtitles may also be defined [**/STITLE**].

This command is valid in any processor.

Menu Paths

Utility Menu>File>Change Title

/PREP7

/PREP7
Enters the model creation preprocessor.

SESSION: Processor Entry
PREP7: Database
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Notes

Enters the general input data preprocessor (PREP7).

This command is valid only at the Begin Level.

Menu Paths

Main Menu>Preprocessor

K

K, *NPT*, *X*, *Y*, *Z*
Defines a keypoint.

PREP7: Keypoints
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

NPT
Reference number for keypoint. If zero, the lowest available number is assigned [**NUMSTR**].

X, ***Y***, ***Z***
Keypoint location in the active coordinate system (may be R, θ, Z or R, θ, Φ). If *X* = P, graphical picking is enabled and all other fields (including *NPT*) are ignored (valid only in the GUI).

Notes

Defines a keypoint in the active coordinate system [**CSYS**] for line, area, and volume descriptions. A previously defined keypoint of the same number will be redefined. Keypoints may be redefined only if it is not yet attached to a line or is not yet meshed. Solid modeling in a toroidal system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>In Active CS

Main Menu>Preprocessor>Modeling>Create>Keypoints>On Working

Plane

L

L, *P1*, *P2*, *NDIV*, *SPACE*, *XV1*, *YV1*, *ZV1*, *XV2*, *YV2*, *ZV2*
Defines a line between two keypoints.

PREP7: Lines
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

P1
Keypoint at the beginning of line. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P2
Keypoint at the end of line.

NDIV
Number of element divisions within this line. Normally this field is not used; specifying divisions with **LESIZE**, etc. is recommended.

SPACE
Spacing ratio. Normally this field is not used, as specifying spacing ratios with the **LESIZE** command is recommended. If positive, *SPACE* is the nominal ratio of the last division size (at *P2*) to the first division size (at *P1*). If the ratio is greater than 1, the division sizes increase from *P1* to *P2*, and if less than 1, they decrease. If *SPACE* is negative, then |*SPACE*| is the nominal ratio of the center division size to those at the ends.

The following fields are used only if specified end slopes on the line are desired, otherwise zero curvature end slopes will be automatically calculated to produce a line which is "straight" in the active coordinate system. To specify end slopes, use the following fields to define a "slope vector" (one for each end of the line, if desired) that has its tail at the origin and its head at the point *XV*, *YV*, *ZV* in the active coordinate system [**CSYS**]. The corresponding end slope of the line will

then be parallel to this "slope vector."

XV1*, *YV1*, *ZV1
Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P1* end of the line. The tail of the vector is at the origin of the coordinate system.

XV2*, *YV2*, *ZV2
Location of the head of the "slope vector" corresponding to the slope at the *P2* end of the line.

Notes

Defines a line between two keypoints from *P1* to *P2*. The line shape may be generated as "straight" (in the active coordinate system) or curved. The line shape is invariant with coordinate system after it is generated. Note that solid modeling in a toroidal coordinate system is not recommended. A curved line is limited to 180°. Lines may be redefined only if not yet attached to an area.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>In Active Coord

LARC

LARC, *P1*, *P2*, *PC*, *RAD*
Defines a circular arc.

PREP7: Lines
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

P1
Keypoint at one end of circular arc line. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P2
Keypoint at other end of circular arc line.

PC
Keypoint defining plane of arc and center of curvature side (with positive radius). Must not lie along the straight line from *P1* to *P2*. *PC* need not be at the center of curvature.

RAD
Radius of curvature of the arc. If negative, assume center of curvature side is opposite to that defined by *PC*. If *RAD* is blank, *RAD* will be calculated from a curve fit through *P1*, *PC*, and *P2*.

Notes

Defines a circular arc line from *P1* to *P2*. The line shape is generated as circular, regardless of the active coordinate system. The line shape is invariant with coordinate system after it is generated.

When dealing with a large radius arc (1e3), or if the location of the arc you create is far away from the origin of your coordinate system, anomalies may occur. You can prevent this by creating the arc at a smaller scale, and then scaling the model back to full size (**LSSCALE**).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>By End KPs & Rad

Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>Through 3 KPs

AL

AL, *L1*, *L2*, *L3*, *L4*, *L5*, *L6*, *L7*, *L8*, *L9*, *L10*
Generates an area bounded by previously defined lines.

PREP7: Areas
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

L1*, *L2*, *L3*, . . . , *L10

List of lines defining area. The minimum number of lines is 3. The positive normal of the area is controlled by the direction of *L1* using the right-hand rule. A negative value of *L1* reverses the normal direction. If *L1* = ALL, use all selected lines with *L2* defining the normal (*L3* to *L10* are ignored and *L2* defaults to the lowest numbered selected line). If *L1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *L1*.

Notes

Lines may be input (once each) in any order and must form a simply connected closed curve. If the area is defined with more than four lines, the lines must also lie in the same plane or on a constant coordinate value in the active coordinate system (such as a plane or a cylinder).

Note: Solid modeling in a toroidal coordinate system is not recommended. Areas may be redefined only if not yet attached to a volume.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Lines

/VIEW

/VIEW, *WN*, *XV*, *YV*, *ZV*
Defines the viewing direction for the display.

GRAPHICS: Views
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

WN
Window number (or ALL) to which command applies (defaults to 1).

XV*, *YV*, *ZV
The object is viewed along the line from point *XV*,*YV*,*ZV* (in the global coordinate system) to the global coordinate system origin. For section displays, the cutting plane is assumed to be perpendicular to this line. If *XV* = WP, modify view to be normal to the currently defined working plane. Defaults to (0,0,1).

Command Default

0,0,1 view. The default reference orientation is X-axis horizontal to the right, Y-axis vertical upward, and Z-axis out from the screen (normal). See **/VUP** command to change reference orientation.

Notes

The view line is always normal to the screen. The view is selected by defining a point (in the global Cartesian coordinate system) representing a point along the

viewing line. This point, and the global Cartesian coordinate system origin, define the line along which the object is viewed while looking toward the origin. Any point along the view line may be used, i.e., (1,1,1) and (2,2,2) give the same view. The display orientation may be changed as desired [**/ANGLE**]. The display coordinate system type may be changed (from Cartesian to cylindrical, spherical, toroidal, etc.) with the **DSYS** command.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane

Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Utility Menu>PlotCtrls>View Settings>Viewing Direction

/REPLOT

/REPLOT, *Label*
Automatically reissues the last display command for convenience.

GRAPHICS: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Label

Controls the type of replot.

RESIZE — Issued internally when a graphics window resize occurs (Default).

FAST — Only applicable for 3-D devices that allow a fast redisplay for changes in the view characteristics only.

Notes

Reissues the last display command (**NPLOT**, **EPLOT**, **KPLOT**, **PLNSOL**, **PLVAR**, etc.), along with its parameters, for convenience. The current display specifications are used.

When the last display command is invalid in a particular processor, the use of the **/REPLOT** command is also invalid in that processor. However, if you attempt a **/REPLOT** and the last display command is invalid in the current processor, ANSYS produces an element display [**EPLOT**] instead, *as long as the last display command was* **PLNSOL**, **PLESOL**, *or* **PLDISP**. ANSYS performs this substitution of **/REPLOT** with **EPLOT** for your convenience.

For example, the **PLNSOL** command, which is used to display solution results as continuous contours, is a valid command in the general postprocessor [**/POST1**]. If you issue **PLNSOL** followed by **/REPLOT** while in the general postprocessor, the **/REPLOT** command effectively reissues your earlier **PLNSOL** command, along with its parameters. But if you then exit the general postprocessor, enter the preprocessor [**/PREP7**], and issue the **/REPLOT** command again, ANSYS internally issues **EPLOT** instead. This occurs because **PLNSOL** is not a valid command in the preprocessor.

When you click on one of the buttons on the Pan, Zoom, Rotate dialog box to manipulate the view of a model, the **/REPLOT** command is issued internally. Thus, the substitution of **/REPLOT** with **EPLOT** as described above may occur not only for the **PLNSOL**, **PLESOL**, and **PLDISP** results display commands, but also for operations that you perform with the Pan, Zoom, Rotate dialog box.

/REPLOT will not show boundary conditions if they are only applied to a solid model and the last display command (for example, **EPLOT**) displays the finite element model. To show boundary conditions, the following options are available:

- Issue **/REPLOT** after you issue the **SBCTRAN** command to transfer solid model boundary conditions to the finite element model.
- Issue **/REPLOT** after you issue a solid model display command (for example, **VPLOT**).

This command is valid in any processor (except as noted above).

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane

Main Menu>General Postproc>Path Operations>Delete Path>All Paths

Main Menu>General Postproc>Path Operations>Delete Path>By Name

Main Menu>General Postproc>Path Operations>Plot Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon

Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

Main Menu>Preprocessor>Path Operations>Delete Path>All Paths

Main Menu>Preprocessor>Path Operations>Delete Path>By Name

Main Menu>Preprocessor>Path Operations>Plot Paths

Utility Menu>Plot>Replot

Utility Menu>PlotCtrls>Style>Symmetry Expansion>Expansion by values

---

VDRAG

VDRAG, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NLP1*, *NLP2*, *NLP3*, *NLP4*, *NLP5*, *NLP6*
Generates volumes by dragging an area pattern along a path.

PREP7: Volumes

MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

NA1*, *NA2*, *NA3*, . . . , *NA6
List of areas in the pattern to be dragged (6 maximum if using keyboard entry). If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NA1* = ALL, all selected areas will be swept along the path. A component name may also be substituted for *NA1*.

NLP1*, *NLP2*, *NLP3*, . . . , *NLP6
List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines. To be continuous, adjacent lines must share the connecting keypoint (the end keypoint of one line must also be first keypoint of the next line).

Notes

Generates volumes (and their corresponding keypoints, lines, and areas) by sweeping a given area pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which the path lines are input (*NLP1*, *NLP2*, etc.). If the drag path is a single line (*NLP1*), the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given area pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves. Lines are generated with the same shapes as the given pattern and the path lines.

Keypoint, line, area, and volume numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]). Adjacent lines use a common keypoint, adjacent areas use a common line, and adjacent volumes use a common area. For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

If element attributes have been associated with the input area via the **AATT** command, the opposite area generated by the **VDRAG** operation will also have those attributes (i.e., the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the input areas are meshed or belong to a meshed volume, the area(s) can be extruded to a 3-D mesh. Note that the *NDIV* argument of the **ESIZE** command should be set before extruding the meshed areas. Alternatively, mesh divisions can be specified directly on the drag line(s) (**LESIZE**). See the *Modeling and Meshing Guide* for more information.

You can use the **VDRAG** command to generate 3-D interface element meshes for elements INTER194 and INTER195. When generating interface element meshes using **VDRAG**, you must specify the line divisions to generate one interface element directly on the drag line using the **LESIZE** command. The source area to be extruded becomes the bottom surface of the interface element. Interface elements must be extruded in what will become the element's local x direction, that is, bottom to top.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>Along Lines

VSBV

VSBV, *NV1*, *NV2*, *SEPO*, *KEEP1*, *KEEP2*
Subtracts volumes from volumes.

PREP7: Booleans
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

NV1

Volume (or volumes, if picking is used) to be subtracted from. If ALL, use all selected volumes. Volumes specified in set *NV2* are removed from set *NV1*. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NV1*.

NV2

Volume (or volumes, if picking is used) to subtract. If ALL, use all selected volumes (except those included in the *NV1* argument). A component name may also be substituted for *NV2*.

SEPO

Behavior if the intersection of the *NV1* volumes and the *NV2* volumes is an area or areas:

(blank) — The resulting volumes will share area(s) where they touch.

SEPO — The resulting volumes will have separate, but coincident area (s) where they touch.

KEEP1

Specifies whether *NV1* volumes are to be deleted:

(blank) — Use the setting of *KEEP* on the **BOPTN** command.

DELETE — Delete *NV1* volumes after **VSBV** operation (override **BOPTN** command settings).

KEEP — Keep *NV1* volumes after **VSBV** operation (override **BOPTN**

command settings).

KEEP2

Specifies whether *NV2* volumes are to be deleted:

(blank) — Use the setting of *KEEP* on the **BOPTN** command.

DELETE — Delete *NV2* volumes after **VSBV** operation (override **BOPTN** command settings).

KEEP — Keep *NV2* volumes after **VSBV** operation (override **BOPTN** command settings).

Notes

Generates new volumes by subtracting the regions common to both *NV1* and *NV2* volumes (the intersection) from the *NV1* volumes. The intersection can be a volume(s) or area(s). If the intersection is an area and *SEPO* is blank, the *NV1* volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEPO* is set to SEPO, *NV1* is divided into two unconnected volumes with separate areas where they touch. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **VSBV**,ALL,ALL will have no effect because all the volumes in set *NV1*will have been moved to set *NV2*.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Volumes

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Volumes

Main Menu>Preprocessor>Modeling>Operate>Subract>Volumes

MP

MP, *Lab*, *MAT*, *C0*, *C1*, *C2*, *C3*, *C4*
Defines a linear material property as a constant or a function of temperature.

PREP7: Materials
MP ME ST PR PRN DS DSS FL EM EH DY PP EME MFS
Product Restrictions

Lab

Valid material property label. Applicable labels are listed under "Material Properties" in the input table for each element type in the *Element Reference*. See *Linear Material Properties* in the *Material Reference* for more complete property label definitions:

ALPD — Mass matrix multiplier for damping.

ALPX — Secant coefficients of thermal expansion (also ALPY, ALPZ).

BETD — Stiffness matrix multiplier for damping.

> **Note:** If used in an explicit dynamic analysis, the value corresponds to the percentage of damping in the high frequency domain. For example, 0.1 roughly corresponds to 10% damping in the high frequency domain.

BETX — Coefficient of diffusion expansion (also BETY, BETZ)

C — Specific heat

CPH — Heat coefficient at constant pressure per unit of mass

CREF — Reference concentration (may not be temperature

dependent)

CSAT — Saturated concentration

CTEX — Instantaneous coefficients of thermal expansion (also CTEY, CTEZ)

CVH — Heat coefficient at constant volume per unit of mass

DENS — Mass density.

DMPR — Constant material damping coefficient

DXX — Diffusivity coefficients (also DYY, DZZ)

EMIS — Emissivity.

ENTH — Enthalpy.

EX — Elastic moduli (also EY, EZ)

GXY — Shear moduli (also GYZ, GXZ)

HF — Convection or film coefficient

KXX — Thermal conductivities (also KYY, KZZ)

LSST — Electric loss tangent

LSSM — Magnetic loss tangent

MGXX — Magnetic coercive forces (also MGYY, MGZZ)

MURX — Magnetic relative permeabilities (also MURY, MURZ)

MU — Coefficient of friction

NUXY — Minor Poisson's ratios (also NUYZ, NUXZ)

PERX — Electric relative permittivities (also PERY, PERZ)

Note: If you enter permittivity values less than 1 for SOLID5, PLANE13, or SOLID98, the program interprets the values as absolute permittivity. Values input for PLANE223, SOLID226, or SOLID227 are always interpreted as relative permittivity.

PRXY — Major Poisson's ratios (also PRYZ, PRXZ).

QRATE — Heat generation rate for thermal mass element MASS71. Fraction of plastic work converted to heat (Taylor-Quinney coefficient) for coupled-field elements PLANE223, SOLID226, and SOLID227.

REFT — Reference temperature. Must be defined as a constant; C1 through C4 are ignored.

RH — Hall Coefficient.

RSVX — Electrical resistivities (also RSVY, RSVZ).

SBKX — Seebeck coefficients (also SBKY, SBKZ).

SONC — Sonic velocity.

THSX — Thermal strain (also THSY, THSZ).

VISC — Viscosity.

MAT

Material reference number to be associated with the elements (defaults to the current MAT setting [**MAT**]).

C0

Material property value, or if a property-versus-temperature polynomial is being defined, the constant term in the polynomial. *C0* can also be a table name (%*tabname*%); if *C0* is a table name, *C1* through *C4* are ignored.

C1*, *C2*, *C3*, *C4

Coefficients of the linear, quadratic, cubic, and quartic terms, respectively, in the property-versus-temperature polynomial. Leave blank (or set to zero) for a constant material property.

Notes

MP defines a linear material property as a constant or in terms of a fourth order polynomial as a function of temperature. (See the **TB** command for nonlinear

material property input.) Linear material properties typically require a single substep for solution, whereas nonlinear material properties require multiple substeps; see *Linear Material Properties* in the *Material Reference* for details.

If the constants C1 - C4 are input, the polynomial

$$\text{Property} = C0 + C1(\text{T}) + C2(\text{T})^2 + C3(\text{T})^3 + C4(\text{T})^4$$

is evaluated at discrete temperature points with linear interpolation between points (that is, a piecewise linear representation) and a constant-valued extrapolation beyond the extreme points. First-order properties use two discrete points (±9999°). The **MPTEMP** or **MPTGEN** commands must be used for second and higher order properties to define appropriate temperature steps. To ensure that the number of temperatures defined via the **MPTEMP** and **MPTGEN** commands is minimally sufficient for a reasonable representation of the curve, ANSYS generates an error message if the number is less than N, and a warning message if the number is less than 2N. The value N represents the highest coefficient used; for example, if C3 is nonzero and C4 is zero, a cubic curve is being used which is defined using 4 coefficients so that N = 4.

A polynomial input is not valid in an explicit dynamic analysis or for *Lab* = ALPD and BETD. C1, C2, C3, and C4 are ignored.

The use of tabular material properties (*C0* = %*tabname*%) is available only for FLUID116. Fluid properties can be evaluated as a function of pressure, temperature, velocity, time, and location (independent variables). Use the ***DIM** command to create the table of property values as a function of the independent variables. Then refer to this table via the **MP** command for the property. If using temperature or pressure independent variables, you need to activate the appropriate pressure or temperature degrees of freedom on the element. Tabular material properties are calculated before the first iteration (that is, using initial values [**IC**]). For more information about using table-type array parameters, see the discussion on applying tabular boundary conditions in the *Basic Analysis Guide*.

When defining a reference temperature (**MP**,REFT), you can convert temperature-dependent secant coefficients of thermal expansion (SCTE) data from the definition temperature to the reference temperature. To do so, issue the **MPAMOD** command.

This command is also valid in SOLUTION.

Product Restrictions

Command Option Lab	Available Products
ALPD	MP ME ST <> <> <> <> <> <> <> DY PP EME MFS
ALPX	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
ALPY	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
ALPZ	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
BETD	MP ME ST <> <> <> <> <> <> <> DY PP EME MFS
BETX	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
BETY	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
BETZ	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS

CREF	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
CSAT	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
DXX	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
DYY	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
DZZ	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
C	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
CPH	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
CTEX	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
CTEY	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
CTEZ	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
CVH	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
DENS	MP ME ST PR PRN DS DSS <> <> EH DY PP EME MFS
DMPR	MP ME ST PR PRN <> <> <> <> <> DY PP EME MFS

EMIS	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
ENTH	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
EX	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
EY	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
EZ	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
GXY	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
GXZ	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
GYZ	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
HF	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
KXX	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
KYY	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
KZZ	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
LSSM	MP <> <> <> <> <> <> <> EM EH <> PP EME <>

LSST	MP ME <> <> <> <> <> <> EM EH <> PP EME MFS
MGXX	MP ME <> <> <> <> <> <> EM <> DY PP EME MFS
MGYY	MP ME <> <> <> <> <> <> EM <> DY PP EME MFS
MGZZ	MP ME <> <> <> <> <> <> EM <> DY PP EME MFS
MU	MP ME ST <> PRN <> <> <> <> <> DY PP EME MFS
MURX	MP ME ST PR PRN DS DSS FL EM EH DY PP EME MFS
MURY	MP ME ST PR PRN DS DSS FL EM EH DY PP EME MFS
MURZ	MP ME ST PR PRN DS DSS FL EM EH DY PP EME MFS
NUXY	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
NUXZ	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
NUYZ	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
PERX	MP ME <> <> <> <> <> FL EM EH DY PP EME MFS
PERY	MP ME <> <> <> <> <> FL EM EH DY PP EME MFS

PERZ	MP ME <> <> <> <> <> FL EM EH DY PP EME MFS
PRXY	MP ME ST PR PRN DS DSS FL <> <> DY PP EME MFS
PRXZ	MP ME ST PR PRN DS DSS FL <> <> DY PP EME MFS
PRYZ	MP ME ST PR PRN DS DSS FL <> <> DY PP EME MFS
QRAT	MP ME ST PR PRN DS DSS <> EM <> DY PP EME MFS
REFT	MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS
RH	MP <> <> <> <> <> <> <> EM <> <> PP EME <>
RSVX	MP ME ST PR PRN DS DSS FL EM EH DY PP EME MFS
RSVY	MP ME ST PR PRN DS DSS FL EM EH DY PP EME MFS
RSVZ	MP ME ST PR PRN DS DSS FL EM EH DY PP EME MFS
SBKX	MP ME <> <> <> <> <> <> EM <> <> PP EME MFS
SBKY	MP ME <> <> <> <> <> <> EM <> <> PP EME MFS
SBKZ	MP ME <> <> <> <> <> <> EM <> <> PP EME MFS

SONC	MP ME <> <> <> <> <> <> <> <> <> PP EME MFS
THSX	MP ME ST <> <> <> <> FL <> <> DY PP EME MFS
THSY	MP ME ST <> <> <> <> FL <> <> DY PP EME MFS
THSZ	MP ME ST <> <> <> <> FL <> <> DY PP EME MFS
VISC	MP ME ST <> <> <> <> FL <> <> DY PP EME MFS

ANSYS Professional NLS supports this command for the coefficient of friction material property (**MP**,MU).

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

ET

ET, *ITYPE*, *Ename*, *KOP1*, *KOP2*, *KOP3*, *KOP4*, *KOP5*, *KOP6*, *INOPR*
Defines a local element type from the element library.

PREP7: Element Type
MP ME ST PR PRN DS DSS FL EM EH DY PP EME MFS

ITYPE
Arbitrary local element type number. Defaults to 1 + current maximum.

Ename
Element name (or number) as given in the element library in Chapter 4 of the *Element Reference*. The name consists of a category prefix and a unique number, such as PIPE288. The category prefix of the name (PIPE for the example) may be omitted but is displayed upon output for clarity. If *Ename* = 0, the element is defined as a null element.

KOP1*, *KOP2*, *KOP3*, . . . , *KOP6
KEYOPT values (1 through 6) for this element, as described in the *Element Reference*.

INOPR
If 1, suppress all element solution printout for this element type.

Notes

The **ET** command selects an element type from the element library and establishes it as a local element type for the current model. Information derived from the element type is used for subsequent commands, so the **ET** command(s) should be issued early. (The *Element Reference* describes the available elements.)

A special option, *Ename* = 0, permits the specified element type to be ignored during solution without actually removing the element from the model. *Ename* may be set to zero only after the element type has been previously defined with a nonzero *Ename*. The preferred method of ignoring elements is to use the select commands (such as **ESEL**).

KOPn are element option keys. These keys (referred to as KEYOPT(*n*)) are used to turn on certain element options for this element. These options are listed under "KEYOPT" in the input table for each element type in the *Element Reference*. KEYOPT values include stiffness formulation options, printout controls, and various other element options. If KEYOPT(7) or greater is needed, input their values with the **KEYOPT** command.

The **ET** command only defines an element type local to your model (from the types in the element library). The **TYPE** or similar [**KATT**, **LATT**, **AATT**, or **VATT**] command must be used to point to the desired local element type before meshing.

To activate the ANSYS program's LS-DYNA explicit dynamic analysis capability, use the **ET** command or its GUI equivalent to choose an element that works only with LS-DYNA (such as SHELL163). Choosing LS-DYNA in the Preferences dialog box does *not* activate LS-DYNA; it simply makes items and options related to LS-DYNA accessible in the GUI.

Menu Paths

Main Menu>Preprocessor>Element Type>Add/Edit/Delete

ESIZE

ESIZE, *SIZE*, *NDIV*
Specifies the default number of line divisions.

PREP7: Meshing
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

SIZE
Default element edge length on surface boundaries (i.e., lines). Divisions are automatically calculated (rounded upward to next integer) from line lengths. If *SIZE* is zero (or blank), use *NDIV*.

NDIV
Default number of element divisions along region boundary lines. Not used if *SIZE* is input.

Notes

Specifies the default number of line divisions (elements) to be generated along the region boundary lines. The number of divisions may be defined directly or automatically calculated. Divisions defined directly for any line [**LESIZE**, **KESIZE**, etc.] are retained. For adjacent regions, the divisions assigned to the common line for one region are also used for the adjacent region. See the **MOPT** command for additional meshing options.

For free meshing operations, if smart element sizing is being used [**SMRTSIZE**] and **ESIZE**,*SIZE* has been specified, *SIZE* will be used as a starting element size, but will be overridden (i.e., a smaller size may be used) to accommodate curvature

and small features.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Size

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Adv Opts

VMESH

VMESH, *NV1*, *NV2*, *NINC*
Generates nodes and volume elements within volumes.

PREP7: Meshing
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

NV1*, *NV2*, *NINC

Mesh volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and all selected volumes [**VSEL**] are meshed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

Notes

Missing nodes required for the generated elements are created and assigned the lowest available numbers [**NUMSTR**]. During a batch run and if elements already exist, a mesh abort will write an alternative database file (**File.DBE**) for possible recovery.

Tetrahedral mesh expansion [**MOPT**,TETEXPND,*Value*] is supported for both the **VMESH** and **FVMESH** commands.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Free

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided

DA

DA, *AREA*, *Lab*, *Value1*, *Value2*
Defines DOF constraints on areas.

SOLUTION: Solid Constraints
MP ME ST PR PRN <> <> <> EM EH <> PP EME MFS

AREA

Area on which constraints are to be specified. If ALL, apply to all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *AREA*.

Lab

Symmetry label (see *2* below):

SYMM — Generate symmetry constraints for non-FLOTRAN models. Requires no *Value1* or *Value2*.

ASYM — Generate antisymmetry constraints for non-FLOTRAN models. Requires no *Value1* or *Value2*.

ANSYS DOF labels (see *1*, *2*, and *3* below):

UX — Displacement in X direction.

UY — Displacement in Y direction.

UZ — Displacement in Z direction.

ROTX — Rotation about X axis.

ROTY — Rotation about Y axis.

ROTZ — Rotation about Z axis.

HDSP — Hydrostatic pressure.

TEMP, TBOT, TE2, TE3, . . ., TTOP — Temperature.

MAG — Magnetic scalar potential.

VOLT — Electric scalar potential.

AX — Magnetic vector potential in X direction (see *4*).

AY — Magnetic vector potential in Y direction.

AZ — Magnetic vector potential in Z direction (see *1*).

CONC — Concentration.

ALL — Applies all appropriate DOF labels *except* HDSP.

FLOTRAN Standard DOF Labels (see *7*): VX, VY, VZ, PRES, TEMP, ENKE, ENDS
FLOTRAN Species Labels (See *8*): SP01, SP02, SP03, SP04, SP05, SP06
FLOTRAN Arbitrary Lagrangian-Eulerian formulation Mesh Displacement Labels (See *9*): UX, UY, UZ

Value1

Value of DOF or table name reference on the area. Valid for all DOF labels. To specify a table, enclose the table name in % signs (e.g., **DA**,*AREA*, TEMP,%*tabname*%). Use the ***DIM** command to define a table.
If *Lab* = ENKE and *Value1* = -1, a FLOTRAN flag is set to indicate a moving wall.
If *Lab* = ENDS and *Value1* = -1, FLOTRAN generalized symmetry conditions are applied. Velocity components are set tangential to the symmetry surface if the ALE formulation is not activated. They are set equal to the mesh velocity if the ALE formulation is activated.

Value2

For FLOTRAN DOFs:

0 — Values are applied only to nodes within the area.

1 — Values are applied to the edges of the area as well as to the internal

nodes. (See *Z*.)

For MAG and VOLT DOFs:

Value of the imaginary component of the degree of freedom.

Notes

1. For elements SOLID236 and SOLID237, if *Lab* = AZ and *Value1* = 0, this sets the flux-parallel condition for the edge formulation. (A flux-normal condition is the natural boundary condition.) Do *not* use the **DA** command to set the edge-flux DOF, AZ to a nonzero value.
2. If *Lab* = MAG and *Value1* = 0, this sets the flux-normal condition for the magnetic scalar potential formulations (MSP) (A flux-parallel condition is the natural boundary condition for MSP.)
3. If *Lab* = VOLT and *Value1* = 0, the J-normal condition is set (current density (J) flow normal to the area). (A J-parallel condition is the natural boundary condition.)
4. For elements HF119 and HF120, used in high-frequency electromagnetic analysis, the AX DOF is not an x-component of a vector potential, but rather a tangential component of E (the electric field) on the element edges and faces. To specify an Electric Wall condition, set AX to zero. For more information, see the *High-Frequency Electromagnetic Analysis Guide*.
5. You can transfer constraints from areas to nodes with the **DTRAN** or **SBCTRAN** commands. See the **DK** command for information about generating other constraints on areas for non-FLOTRAN models.
6. Symmetry and antisymmetry constraints are generated as described for the **DSYM** command.
7. For the velocity DOF (VX, VY, VZ), a zero value will override a nonzero value at the intersection of two areas.
8. You can use the **MSSPEC** command to change FLOTRAN species labels to user-defined labels. You must define these labels with the **MSSPEC** command before using them on the **DA** command.

9. Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following degree of freedom labels: Electric (VOLT), FLOTRAN (UX, UY, UZ, PRES, VX, VY, VZ, ENKE, ENDS, TEMP, SP01, SP02, SP03, SP04, SP05, and SP06); Structural (UX, UY, UZ, ROTX, ROTY, ROTZ), and temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP).
10. Constraints specified by the **DA** command can conflict with other specified constraints. See Resolution of Conflicting Constraint Specifications\ in the *Basic Analysis Guide* for details.
11. The **DA** command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Boundary>PerfEC>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Boundary>TimeInt>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Boundary>Voltage>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Magnetic>Boundary>ScalarPot>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Areas

**Main Menu>Preprocessor>Loads>Define
Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Areas**

**Main Menu>Preprocessor>Loads>Define
Loads>Apply>Magnetic>Boundary>VectorPot>On Areas**

**Main Menu>Preprocessor>Loads>Define
Loads>Apply>Structural>Displacement>Antisymm B.C.>On Areas**

**Main Menu>Preprocessor>Loads>Define
Loads>Apply>Structural>Displacement>On Areas**

**Main Menu>Preprocessor>Loads>Define
Loads>Apply>Structural>Displacement>Symmetry B.C.>On Areas**

**Main Menu>Preprocessor>Loads>Define
Loads>Apply>Thermal>Temperature>On Areas**

Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>On Areas

Main Menu>Solution>Constraints>Apply>On Areas

**Main Menu>Solution>Define
Loads>Apply>Electric>Boundary>PerfEC>On Areas**

**Main Menu>Solution>Define
Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Areas**

**Main Menu>Solution>Define
Loads>Apply>Electric>Boundary>TimeInt>On Areas**

**Main Menu>Solution>Define
Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Areas**

**Main Menu>Solution>Define
Loads>Apply>Electric>Boundary>Voltage>On Areas**

**Main Menu>Solution>Define
Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Areas**

**Main Menu>Solution>Define
Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Areas**

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Areas

---

SFA

SFA, *AREA*, *LKEY*, *Lab*, *VALUE*, *VALUE2*
Specifies surface loads on the selected areas.

SOLUTION: Solid Surface Loads
MP ME ST PR PRN <> <> <> EM EH <> PP EME MFS

AREA

Area to which surface load applies. If ALL, apply load to all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component may be substituted for *AREA*.

LKEY

Load key associated with surface load (defaults to 1). Load keys (1,2,3, etc.) are listed under "Surface Loads" in the input data table for each element type in the *Element Reference*. *LKEY* is ignored if the area is the face of a volume region meshed with volume elements.

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each area type in the *Element Reference*.

Discipline	Body Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[*1*]	convection
	HFLUX[*1*]	heat flux
	RAD	radiation

	RDSF	surface-to-surface radiation
Acoustic fluid	FSI[*2*]	fluid-structure interaction flag
	IMPD	impedance boundary flag
	SHLD	surface normal velocity or acceleration
	MXWF	Maxwell surface flag or equivalent source surface
	FREE	free surface flag
	INF	exterior Robin radiation boundary flag
	CONV	Attenuation coefficient
Magnetic	MXWF	Maxwell force flag
	MCI	magnetic circuit interface
Electric	CHRGS	surface charge density
	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for INFIN110 and INFIN111
High-frequency electromagnetic	PORT	number 1-50 for a waveguide exterior port
	SHLD	surface shielding properties
	IMPD	surface impedance
Field-surface interface	FSIN[*3*]	field-surface interface number

1. Thermal labels CONV and HFLUX are mutually exclusive.

2. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID129 or FLUID130 elements.
3. For a fluid-solid interaction analysis, apply the field-surface interface flag (Label = FSIN) twice: once for the fluid side (FLUID141 or FLUID142 elements) and once for the solid side.

VALUE

Surface load value or table name reference for specifying tabular boundary conditions.

If *Lab* = CONV, *VALUE* is typically the film coefficient and *VALUE2* (below) is typically the bulk temperature. If *Lab* = CONV and *VALUE* = -*N*, the film coefficient may be a function of temperature and is determined from the HF property table for material *N* [**MP**]. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user-defined for some elements. For acoustic harmonic and transient analyses, *VALUE* is the attenuation coefficient of the surface.

If *Lab* = MCI, VALUE indicates current direction (-1; current flow into the element face (IN), +1; current flow out of the element face (OUT).

If *Lab* = RAD, *VALUE* is the surface emissivity.

If *Lab* = PORT, *VALUE* is a port number representing a waveguide exterior port. The port number must be an integer between 1 and 50.

If *Lab* = SHLD, *VALUE* is surface conductivity for high-frequency electromagnetics. For acoustics, *VALUE* is the surface normal velocity in harmonic analysis and the surface normal acceleration in transient analysis.

If *Lab* = IMPD, *VALUE* is resistance in ohms/square for high-frequency electromagnetics. For acoustic harmonic response analyses, *VALUE* is resistance in (N)(s)/m^3 if *VALUE* > 0 and is conductance in mho if *VALUE* < 0. In acoustic transient analyses, VALUE2 is not used.

If *Lab* = RDSF, *VALUE* is the emissivity value; the following conditions apply: If *VALUE* is between 0 and 1, apply a single value to the surface. If *VALUE* = -*N*, the emissivity may be a function of the temperature, and is determined from the EMISS property table for material *N* (**MP**). The material *N* does not need to correlate with the underlying solid thermal

elements.
If *Lab* = FSIN in a Multi-field solver (single or multiple code coupling) analysis, *VALUE* is the surface interface number and *LKEY* is ignored. If *Lab* = FSIN in a unidirectional ANSYS to CFX analysis, *VALUE* is not used unless the ANSYS analysis is performed using the Multi-field solver.

VALUE2

Second surface load value (if any).
If *Lab* = CONV, *VALUE2* is typically the bulk temperature for thermal analyses. For acoustic analyses, VALUE2 is not used. .
If *Lab* = RAD*VALUE2* is ambient temperature.
If *Lab* = SHLD, *VALUE2* is relative permeability (defaults to 1.0) for high-frequency electromagnetics. For acoustic analyses, VALUE2 is the phase angle of the normal surface velocity (defaults to zero) for harmonic response analyses while VALUE2 is not used for transient analyses.
If *Lab* = IMPD, *VALUE2* is reactance in ohms/square for high-frequency electromagnetics. For acoustics, *VALUE2* is reactance in (N)(s)/m^3 if *VALUE* > 0 and is the product of susceptance and angular frequency if *VALUE* < 0.
If *Lab* = RDSF, *VALUE2* is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open, radiation will also occur to ambient. If *VALUE2* is negative radiation direction is reversed and will occur inside the element for the flagged radiation surfaces. Negative value of enclosure number is applicable for FLUID141 and FLUID142 elements, to model radiation occurring between surfaces inside the fluid domain.
If *Lab* = FSIN in a unidirectional ANSYS to CFX analysis, *VALUE2* is the surface interface number (not available from within the GUI).

Notes

Surface loads may be transferred from areas to elements with the **SFTRAN** or **SBCTRAN** commands. See the **SFGRAD** command for an alternate tapered load capability.

Tabular boundary conditions (*VALUE* = %*tabname*% and/or *VALUE2* = %*tabname*%) are available for the following surface load labels (*Lab*) only: PRES (real and/or imaginary components), CONV (film coefficient and/or bulk temperature) or HFLUX, and RAD (surface emissivity and ambient temperature). Use the ***DIM** command to define a table.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Boundary>AppImped_E>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Boundary>AppShield>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Excitation>AppSurfChar>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Flag>AppInfinite>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Electric>Flag>AppMaxwell>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Fluid/ANSYS>Field Surface>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Fluid/ANSYS>Impedance>On Areas

Main Menu>Preprocessor>Loads>Define
Loads>Apply>Magnetic>Flag>AppInfinite>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppMCI>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Surface Rad>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppImped_E>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppShield>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Areas

Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Areas

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field

Surface>On Areas

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppMCI>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Surface Rad>On Areas

/SOLU

/SOLU
Enters the solution processor.

SESSION: Processor Entry
SOLUTION: Analysis Options
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Notes

This command is valid only at the Begin Level.

Menu Paths

Main Menu>Solution

SOLVE

SOLVE, *Action*
Starts a solution.

SOLUTION: Analysis Options
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Action

Action to be performed on solve (used only for linear perturbation analyses).

ELFORM — Reform all appropriate element matrices in the first phase of a linear perturbation analysis.

Notes

Starts the solution of one load step of a solution sequence based on the current analysis type and option settings. Use *Action* = ELFORM only in the first phase of a linear perturbation analysis.

Menu Paths

Main Menu>Drop Test>Solve

Main Menu>Solution>Run FLOTRAN

Main Menu>Solution>Solve

Main Menu>Solution>Solve>Current LS

FINISH

FINISH
Exits normally from a processor.

SESSION: Processor Entry
DISPLAY: Action
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Notes

Exits any of the ANSYS processors or the DISPLAY program. For the ANSYS processors, data will remain intact in the database but the database is not automatically written to a file (use the **SAVE** command to write the database to a file). See also the **/QUIT** command for an alternate processor exit command. If exiting POST1, POST26, or OPT, see additional notes below.

POST1: Data in the database will remain intact, including the POST1 element table data, the path table data, the fatigue table data, and the load case pointers.

POST26: Data in the database will remain intact, except that POST26 variables are erased and specification commands (such as **FILE**, **PRTIME**, **NPRINT**, etc.) are reset. Use the **/QUIT** command to exit the processor and bypass these exceptions.

This command is valid in any processor. This command is not valid at the Begin level.

Menu Paths

Main Menu>Finish

/POST1

/POST1
Enters the database results postprocessor.

SESSION: Processor Entry
POST1: Set Up
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Notes

Enters the general database results postprocessor (POST1). All load symbols (**/PBC**, **/PSF**, or **/PBF**) are automatically turned off with this command.

This command is valid only at the Begin Level.

Menu Paths

Main Menu>General Postproc

/DSCALE

/DSCALE, *WN*, *DMULT*
Sets the displacement multiplier for displacement displays.

GRAPHICS: Scaling
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

WN
Window number (or ALL) to which command applies (defaults to 1).

DMULT

AUTO or 0 — Scale displacements automatically so that maximum displacement (vector amplitude) displays as 5 percent of the maximum model length, as measured in the global Cartesian X, Y, or Z directions.

1 — Do not scale displacements (i.e., scale displacements by 1.0, true to geometry). Often used with large deflection results.

FACTOR — Scale displacements by numerical value input for FACTOR.

OFF — Remove displacement scaling (i.e., scale displacements by 0.0, no distortion).

USER — Set *DMULT* to that used for last display (useful when last *DMULT* value was automatically calculated).

Command Default

The default value is AUTO or 0 except when:

Large deflection effects are included (**NLGEOM**,ON) and it is not a modal analysis; then the default is 1.

It is a spectrum analysis (**ANTYPE**,SPECTR); then the default is OFF.

The amplitude of a time-harmonic solution is computed using the **HRCPLX** command (OMEGAT ≥ 360°); then the default is OFF.

The amplitude of a complex modal or harmonic solution is stored into the database using the **SET** command (*KIMG* = AMPL); then the default is OFF.

Notes

If Multi-Plots are not being displayed, and the current device is a 3-D device [**/SHOW**,3D], then the displacement scale in all active windows will be the same, even if separate **/DSCALE** commands are issued for each active window. For efficiency, ANSYS 3-D graphics logic maintains a single data structure (segment), which contains only one displacement scale. The program displays the same segment (displacement scale) in all windows. Only the view settings will be different in each of the active windows.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Displacement Scaling

PLDISP

PLDISP, *KUND*
Displays the displaced structure.

POST1: Results
MP ME ST PR PRN DS DSS <> <> <> DY PP EME MFS

KUND

Undisplaced shape key:

- **0** — Display only displaced structure.
- **1** — Overlay displaced display with similar undisplaced display (appearance is system-dependent).
- **2** — Same as 1 except overlay with undisplaced edge display (appearance is system-dependent).

Notes

Displays the displaced structure for the selected elements.

For information on true scale plots, refer to the description of the **/DSCALE** command [**/DSCALE**,,1.0].

Menu Paths

Main Menu>General Postproc>Plot Results>Deformed Shape

Utility Menu>Plot>Results>Deformed Shape
Utility Menu>PlotCtrls>Animate>Deformed Shape

PLNSOL

PLNSOL, *Item*, *Comp*, *KUND*, *Fact*, *FileID*
Displays results as continuous contours.

POST1: Results
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Item

Label identifying the item. Valid item labels are shown in *Table 238: PLNSOL - Valid Item and Component Labels* below. Some items also require a component label.

Comp

Component of the item (if required). Valid component labels are shown in *Table 238: PLNSOL - Valid Item and Component Labels* below.

KUND

Undisplaced shape key:

- **0** — Do not overlay undeformed structure display
- **1** — Overlay displaced contour plot with undeformed display (appearance is system-dependent)
- **2** — Overlay displaced contour plot with undeformed edge display (appearance is system-dependent)

Fact

Scale factor for 2-D display for contact items. Default value is 1. A negative scaling factor may be used to invert the display.

FileID

The file index number (obtained via the **NLDIAG**,NRRE,ON command). Valid only for *Item* = NRRE.

Notes

Displays the solution results as continuous contours across element boundaries for the selected nodes and elements. For example, **PLNSOL**,S,X displays the X component of stress S (that is, the SX stress component). Various element results depend upon the recalculation method and the selected results location [**AVPRIN**, **RSYS**, **LAYER**, **SHELL**, and **NSEL**]. Contours are determined by linear interpolation within each element from the nodal values, which are averaged at a node whenever two or more elements connect to the same node (except for FMAG, which is summed at the node).

For PowerGraphics displays [***/GRAPHICS***,POWER], results are plotted only for the model exterior surface. The items marked with [*2*] are not supported by PowerGraphics. To plot midside nodes, you must first issue ***/EFACET***,2.

Table 238: PLNSOL - Valid Item and Component Labels

General Item and Component Labels PLNSOL, *Lab, Item, Comp*

Item	Comp	Description
Valid item and component labels for nodal degree of freedom results are:		
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum
TEMP [*1*]		Temperature
PRES		Pressure
VOLT		Electric potential
MAG		Magnetic scalar potential
CONC		Concentration [*2*]

V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum in a fluid analysis, or X, Y, or Z velocity or vector sum in an ANSYS LS-DYNA analysis.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum in an electromagnetic analysis, or X, Y, or Z acceleration or vector sum in an ANSYS LS-DYNA analysis.
VEL	X, Y, Z, SUM	X, Y, or Z velocity or vector sum in a structural transient dynamic analysis (**ANTYPE**,TRANS).
ACC	X, Y, Z, SUM	X, Y, or Z acceleration or vector sum in a structural transient dynamic analysis (**ANTYPE**,TRANS).
OMG	X, Y, Z, SUM	X, Y, or Z rotational velocity or vector sum in a structural transient dynamic analysis (**ANTYPE**, TRANS).
DMG	X, Y, Z, SUM	X, Y, or Z rotational acceleration or vector sum in a structural transient dynamic analysis (**ANTYPE**,TRANS).
ENKE		Turbulent kinetic energy (FLOTRAN)
ENDS		Turbulent energy dissipation (FLOTRAN)
SP0*n*		Mass fraction of species *n*, where *n* = 1 to 6 (FLOTRAN). If a species is given a user-defined name [**MSSPEC**], use that name instead of *n*.
WARP		Warping
NRRE	FX, FY, FZ, FNRM, MX, MY, MZ, MNRM	Plot the Newton-Raphson residuals from the file you obtained via the **NLDIAG**,NRRE,ON command. The FNRM and MNRM labels are computed as the square root of the sum of the squares of the residual component forces or moments (FX,FY,FZ, MX, MY, MZ).

		[5]
		When KUND = 0, use the absolute value of the residual from the files (default).
SPL		Sound pressure level.

Valid item and component labels for element results are:

	X, Y, Z, XY, YZ, XZ	Component stress
S	1, 2, 3	Principal stress
	INT	Stress intensity
	EQV	Equivalent stress
	X, Y, Z, XY, YZ, XZ	Component elastic strain
EPEL	1, 2, 3	Principal elastic strain
	INT	Elastic strain intensity
	EQV	Elastic equivalent strain
	X, Y, Z, XY, YZ, XZ	Component thermal strain
EPTH	1, 2, 3	Principal thermal strain
	INT	Thermal strain intensity
	EQV	Thermal equivalent strain
	X, Y, Z, XY, YZ, XZ	Component plastic strain
EPPL	1, 2, 3	Principal plastic strain
	INT	Plastic strain intensity
	EQV	Plastic equivalent strain
	X, Y, Z, XY, YZ, XZ	Component creep strain

EPCR	1, 2, 3	Principal creep strain
	INT	Creep strain intensity
	EQV	Creep equivalent strain
EPSW		Swelling strain
	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (EPEL + EPPL + EPCR)
EPTO	1, 2, 3	Principal total mechanical strain
	INT	Total mechanical strain intensity
	EQV	Total mechanical equivalent strain
	X, Y, Z, XY, YZ, XZ	Component total mechanical *and* thermal strain (EPEL + EPPL + EPCR + EPTH)
EPTT	1, 2, 3	Principal total mechanical *and* thermal strain
	INT	Total mechanical *and* thermal strain intensity
	EQV	Total mechanical *and* thermal equivalent strain
	X, Y, Z, XY, YZ, XZ	Component diffusion strain [*2*]
EPDI	1, 2, 3	Principal diffusion strain
	INT	Diffusion strain intensity
	EQV	Diffusion equivalent strain
	X,Y,Z,XY,YZ, ZX	Components of Biot's effective stress.
ESIG	1, 2, 3	Principal stresses of Biot's effective stress.
	INT	Stress intensity of Biot's effective stress.
	EQV	Equivalent stress of Biot's effective stress.
	TPOR	Total porosity (Gurson material model).
DPAR	GPOR	Porosity due to void growth.

	NPOR	Porosity due to void nucleation.
	SEPL	Equivalent stress (from stress-strain curve)
	SRAT	Stress state ratio
	HPRES	Hydrostatic pressure
NL	EPEQ	Accumulated equivalent plastic strain
	CREQ	Accumulated equivalent creep strain
	PSV	Plastic state variable
	PLWK	Plastic work/volume
	ELASTIC	Elastic strain energy density
SEND	PLASTIC	Plastic strain energy density
	CREEP	Creep strain energy density
CDM	DMG	Damage variable
	LM	Maximum previous strain energy for virgin material
	MAX	Maximum of all active failure criteria defined at the current location. (See the **FCTYP** command for details.) [*2*][*6*]
	EMAX	Maximum Strain Failure Criterion [*2*][*6*]
	SMAX	Maximum Stress Failure Criterion [*2*][*6*]
	TWSI	Tsai-Wu Strength Index Failure Criterion [*2*][*6*]
	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion [*2*][*6*]
	HFIB	Hashin Fiber Failure Criterion. [*2*][*6*][*8*]
FAIL	HMAT	Hashin Matrix Failure Criterion. [*2*][*6*][*8*]
	PFIB	Puck Fiber Failure Criterion. [*2*][*6*][*8*]
	PMAT	Puck Matrix Failure Criterion. [*2*][*6*][*8*]
	L3FB	LaRc03 Fiber Failure Criterion. [*2*][*6*][*8*]

	L3MT	LaRc03 Matrix Failure Criterion. [2][6][8]
	L4FB	LaRc04 Fiber Failure Criterion. [2][6][8]
	L4MT	LaRc04 Matrix Failure Criterion. [2][6][8]
	USR1, USR2, ..., USR9	User-defined failure criteria [2][6][7][8]
PFC		Failure criteria based on the effective stresses in the damaged material. **Components:** Maximum of all failure criteria defined at current location (MAX), fiber tensile failure (FT), fiber compressive failure (FC), matrix tensile failure (MT), and matrix compressive (MC).
PDMG		Progressive damage parameters. **Components:** Damage status (STAT, 0--undamaged, 1--damaged), fiber tensile damage variable (FT), fiber compressive damage variable (FC), matrix tensile damage variable (MT), matrix compressive damage variable (MC), shear damage variable (S), and energy dissipated per unit volume (SED).
SVAR	1, 2, 3, ... N	State variable [2]
GKS	X, XY, XZ	Gasket component stress
GKD	X, XY, XZ	Gasket component total closure
GKDI	X, XY, XZ	Gasket component total inelastic closure
GKTH	X, XY, XZ	Gasket component thermal closure
SS	X, XY, XZ	Interface traction (stress)
SD	X, XY, XZ	Interface separation
FICT	TEMP	Fictive temperature

For contact results PowerGraphics is applicable for 3-D models only.

CONT [3]	STAT	Contact status[4]: 3-closed and sticking 2-closed and sliding 1-open but near contact 0-open and not near contact
	PENE	Contact penetration
	PRES	Contact pressure
	SFRIC	Contact friction stress
	STOT	Contact total stress (pressure plus friction)
	SLIDE	Contact sliding distance
	GAP	Contact gap distance
	FLUX	Total heat flux at contact surface
	CNOS	Total number of contact status changes during substep.
	FPRS	Fluid penetration pressure
TG	X, Y, Z, SUM	Component thermal gradient or vector sum
TF	X, Y, Z, SUM	Component thermal flux or vector sum
PG	X, Y, Z, SUM	Component pressure gradient or vector sum
EF	X, Y, Z, SUM	Component electric field or vector sum
D	X, Y, Z, SUM	Component electric flux density or vector sum
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum
B	X, Y, Z, SUM	Component magnetic flux density or vector sum
CG	X, Y, Z, SUM	Component concentration gradient or vector sum [2]
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum [2]
FMAG	X, Y, Z, SUM	Component electromagnetic force or vector sum [2].
		Conduction current density for elements that support

JC	X, Y, Z, SUM	conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [2].
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).

Valid item labels for FLOTRAN nodal results are:

TTOT	Total temperature
HFLU	Heat flux
HFLM	Heat transfer (film) coefficient
COND	Fluid laminar conductivity
PCOE	Pressure coefficient
PTOT	Total (stagnation) pressure
MACH	Mach number
STRM	Stream function (2-D applications only)
DENS	Fluid density
VISC	Fluid laminar viscosity
SPHT	Specific heat [2]
EVIS	Fluid effective viscosity
CMUV	Turbulent viscosity coefficient
ECON	Fluid effective conductivity
YPLU	Y+, a turbulent law of the wall parameter
TAUW	Shear stress at the wall
SFTS	Surface tension coefficient
LMD*n*	Laminar mass diffusion coefficient for species *n*, where *n* = 1 to 6
EMD*n*	Effective mass diffusion coefficient for species *n*, where *n* = 1 to 6

RDFL Radiation heat flux [*2*]

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP to view the individual temperature degree of freedom. When other thermal elements are included in the model, they should be unselected to avoid plotting undefined information. To view all temperatures in the same plot, set **/ESHAPE**,1 and **/GRAPHICS**,POWER and issue **PLNSOL**,TEMP.
2. Not supported by PowerGraphics
3. For the CONT items for elements CONTA171 through CONTA177, the reported data is averaged across the element. To obtain a more meaningful STAT value, use the **PLESOL** command.
4. For MPC-based contact definitions, the value of STAT can be negative. This indicates that one or more contact constraints were intentionally removed to prevent overconstraint. STAT = -3 is used for MPC bonded contact; STAT = -2 is used for MPC no-separation contact.
5. When plotting Newton-Raphson residual items (*Item* = NRRE) from a file on the deformed geometry, the displacements are based on the current set of results in the database. These displacements may not correspond to the loadstep and substep in the **.nr***xxxxx* file. (For more information about **.nr***xxxxx* files and nonlinear diagnostics postprocessing, see the description of the **NLDPOST** command and *Performing Nonlinear Diagnostics* .)
6. Works only if failure criteria information is provided. (For more information, see the documentation for the **FC** and **TB** commands.)
7. Works only if user failure criteria routine is provided.
8. Must be added via the **FCTYP** command first.

Menu Paths

Main Menu>Drop Test>Animate Results

Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu

Utility Menu>Plot>Results>Contour Plot>Nodal Solution

Utility Menu>PlotCtrls>Animate>Animate Over Results

Utility Menu>PlotCtrls>Animate>Animate Over Time

Utility Menu>PlotCtrls>Animate>Deformed Results

Utility Menu>PlotCtrls>Animate>Dynamic Results

Utility Menu>PlotCtrls>Animate>Isosurfaces

Utility Menu>PlotCtrls>Animate>Mode Shape

Utility Menu>PlotCtrls>Animate>Q-Slice Contours

Utility Menu>PlotCtrls>Animate>Time-harmonic

/CTYPE

/CTYPE, *KEY*, *DOTD*, *DOTS*, *DSHP*, *TLEN*
Specifies the type of contour display.

GRAPHICS: Style
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

KEY

Type of display:

0 — Standard contour display.

1 — Isosurface display.

2 — Particle gradient display.

3 — Gradient triad display.

DOTD

Maximum dot density for particle gradient display (*KEY* = 2). Density is expressed as dots per screen width (defaults to 30).

DOTS

Dot size for particle gradient display (*KEY* = 2). Size is expressed as a fraction of the screen width (defaults to 0.0 (single dot width)).

DSHP

Spherical dot shape precision for particle gradient display (*KEY* = 2). (3-D options are supported only on 3-D devices):

0 — Flat 2-D circular dot.

1 — Flat-sided 3-D polyhedron.

n — 3-D sphere with *n* (>1) polygon divisions per 90° of radius.

TLEN

Maximum length of triads for gradient triad display (*KEY* = 3). Value is

expressed as a fraction of the screen width (defaults to 0.067).

Command Default

Standard contour display.

Notes

Use **/CTYPE**,STAT to display the current settings. Only the standard contour display [**/CTYPE**,0) and the isosurface contour display [**/CTYPE**,1] are supported by PowerGraphics [**/GRAPHICS**,POWER].

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Contour Style

/CVAL

/CVAL, *WN*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*, *V7*, *V8*
Specifies nonuniform contour values on stress displays.

GRAPHICS: Labeling
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

WN
Window number (or ALL) to which command applies (defaults to 1).

V1*, *V2*, *V3*, . . . , *V8
Up to 8 contour values may be specified (in ascending order). The 0.0 value (if any) must not be the last value specified. If no values are specified, all contour specifications are erased and contours are automatically calculated.

Command Default

Nine contour values uniformly spaced between the extreme values.

Notes

This command is similar to the **/CONTOUR** command. With **/CVAL**, however, you define the upper level of each contour band instead of having the contours uniformly distributed over the range. The minimum value (including a zero value for the first band) for a contour band cannot be specified. If you use both **/CONTOUR** and **/CVAL**, the last command issued takes precedence.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Non-uniform Contours

PLESOL

PLESOL, *Item*, *Comp*, *KUND*, *Fact*
Displays the solution results as discontinuous element contours.

POST1: Results
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

Item
Label identifying the item. Valid item labels are shown in *Table 237: PLESOL - Valid Item and Component Labels* below. Some items also require a component label.

Comp
Component of the item (if required). Valid component labels are shown in *Table 237: PLESOL - Valid Item and Component Labels* below.

KUND
Undisplaced shape key:

- **0** — Do not overlay undeformed structure display
- **1** — Overlay displaced contour plot with undeformed display (appearance is system-dependent)
- **2** — Overlay displaced contour plot with undeformed edge display (appearance is system-dependent)

Fact
Scale factor for 2-D display of contact items (defaults to 1). A negative scaling factor may be used to invert the display.

Notes

Displays the solution results as element contours discontinuous across element boundaries for the selected elements. For example, **PLESOL**,S,X displays the X component of stress S (that is, the SX stress component). Various element results depend on the calculation method and the selected results location (**AVPRIN**, **RSYS**, and **ESEL**). Contours are determined by linear interpolation within each element, unaffected by the surrounding elements (i.e., no nodal averaging is performed). The discontinuity between contours of adjacent elements is an indication of the gradient across elements. Component results are displayed in the active results coordinate system [**RSYS**] (default is the global Cartesian). See the **ETABLE** and **PLETAB** commands for displaying items not available through this command (such as line element results).

For PowerGraphics displays [***I*GRAPHICS**,POWER], results are plotted only for the model exterior surface. The items marked with [*1*] in *Table 237: PLESOL - Valid Item and Component Labels* are not supported by PowerGraphics.

Table 237: PLESOL - Valid Item and Component Labels

General Item and Component Labels PLESOL, *Lab, Item, Comp*

Item	Comp	Description
		Valid item and component labels for element results are:
S	X, Y, Z, XY, YZ, XZ	Component stress.
	1, 2, 3	Principal stress.
	INT	Stress intensity.
	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
	1, 2, 3	Principal elastic strain.
	INT	Elastic strain intensity.

	EQV	Elastic equivalent strain.
	X, Y, Z, XY, YZ, XZ	Component plastic strain.
EPPL	1, 2, 3	Principal plastic strain.
	INT	Plastic strain intensity.
	EQV	Plastic equivalent strain.
	X, Y, Z, XY, YZ, XZ	Component creep strain.
EPCR	1, 2, 3	Principal creep strain.
	INT	Creep strain intensity.
	EQV	Creep equivalent strain.
	X, Y, Z, XY, YZ, XZ	Component thermal strain.
EPTH	1, 2, 3	Principal thermal strain.
	INT	Thermal strain intensity.
	EQV	Thermal equivalent strain.
EPSW		Swelling strain.
	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (EPEL + EPPL + EPCR).
EPTO	1, 2, 3	Principal total mechanical strain.
	INT	Total mechanical strain intensity.
	EQV	Total mechanical equivalent strain.
	X, Y, Z, XY, YZ, XZ	Total mechanical *and* thermal strain (EPEL + EPPL + EPCR + EPTH). [*8*]
EPTT	1, 2, 3	Principal total mechanical *and* thermal strain.
	INT	Total mechanical *and* thermal strain intensity.

	EQV	Total mechanical *and* thermal equivalent strain.
EPDI	X, Y, Z, XY, YZ, XZ	Component diffusion strain [*1*]
	1, 2, 3	Principal diffusion strain
	INT	Diffusion strain intensity
	EQV	Diffusion equivalent strain
	SEPL	Equivalent stress (from stress-strain curve).
	SRAT	Stress state ratio.
	HPRES	Hydrostatic pressure.
NL	EPEQ	Accumulated equivalent plastic strain.
	CREQ	Accumulated equivalent creep strain.
	PSV	Plastic state variable.
	PLWK	Plastic work/volume.
	ELASTIC	Elastic strain energy density.
SEND	PLASTIC	Plastic strain energy density.
	CREEP	Creep strain energy density.
CDM	DMG	Damage variable
	LM	Maximum previous strain energy for virgin material
	MAX	Maximum of all active failure criteria defined at the current location (See the **FCTYP** command for details.) [*1*][*3*]
	EMAX	Maximum Strain Failure Criterion. [*1*][*3*]
	SMAX	Maximum Stress Failure Criterion. [*1*][*3*]
	TWSI	Tsai-Wu Strength Index Failure Criterion. [*1*][*3*]
	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion. [*1*][*3*]

FAIL	HFIB	Hashin Fiber Failure Criterion. [1][3][5]
	HMAT	Hashin Matrix Failure Criterion. [1][3][5]
	PFIB	Puck Fiber Failure Criterion. [1][3][5]
	PMAT	Puck Matrix Failure Criterion. [1][3][5]
	L3FB	LaRc03 Fiber Failure Criterion. [1][3][5]
	L3MT	LaRc03 Matrix Failure Criterion. [1][3][5]
	L4FB	LaRc04 Fiber Failure Criterion. [1][3][5]
	L4MT	LaRc04 Matrix Failure Criterion. [1][3][5]
	USR1, USR2, ..., USR9	User-defined failure criteria. [1][3][4][5]
PFC		Failure criteria based on the effective stresses in the damaged material. **Components:** Maximum of all failure criteria defined at current location (MAX), fiber tensile failure (FT), fiber compressive failure (FC), matrix tensile failure (MT), and matrix compressive (MC).
PDMG		Progressive damage parameters. **Components:** Damage status (STAT, 0--undamaged, 1--damaged), fiber tensile damage variable (FT), fiber compressive damage variable (FC), matrix tensile damage variable (MT), matrix compressive damage variable (MC), shear damage variable (S), and energy dissipated per unit volume (SED).
	LAY	Layer number where the maximum of all active failure criteria over the entire element occurs. [1][3] Number of the maximum-failure criterion over the entire element [1][3] : 1 - EMAX

FCMX	FC	2 - SMAX 3 - TWSI 4 - TWSR 5 - PFIB 6 - PMAT 7 - HFIB 8 - HMAT 9 - L3FB 10 - L3MT 11 - L4FB 12 - L4MT 13~21 - USR1~USR9
	VAL	Value of the maximum failure criterion over the entire element. [*1*][*3*]
SVAR	1, 2, 3, ... N	State variable. [*1*]
GKS	X, XY, XZ	Gasket component stress.
GKD	X, XY, XZ	Gasket component total closure.
GKDI	X, XY, XZ	Gasket component total inelastic closure.
GKTH	X, XY, XZ	Gasket component thermal closure.
SS	X, XY, XZ	Interface traction (stress)
SD	X, XY, XZ	Interface separation
	STAT	Contact status [*2*]: 3-closed and sticking 2-closed and sliding 1-open but near contact

		0-open and not near contact
CONT	PENE	Contact penetration
	PRES	Contact pressure
	SFRIC	Contact friction stress
	STOT	Contact total stress (pressure plus friction)
	SLIDE	Contact sliding distance
	GAP	Contact gap distance
	FLUX	Total heat flux at contact surface
	CNOS	Total number of contact status changes during substep.
	FPRS	Fluid penetration pressure
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
CG	X, Y, Z, SUM	Component concentration gradient or vector sum [*1*]

DF	X, Y, Z, SUM	Component diffusion flux density or vector sum [*1*]
FMAG	X, Y, Z, SUM	Component electromagnetic force or vector sum [*1*].
P	X, Y, Z, SUM	Pointing vector component or sum [*1*].
SERR [*6*]		Structural error energy [*1*].
SDSG [*6*]		Absolute value of the maximum variation of any nodal stress component [*1*].
TERR [*6*]		Thermal error energy [*1*].
TDSG [*6*]		Absolute value of the maximum variation of any nodal thermal gradient component [*1*].
F	X, Y, Z	X, Y, or Z structural force [*1*][*7*].
M	X, Y, Z	X, Y, or Z structural moment [*1*].
HEAT		Heat flow [*1*].
FLOW		Fluid flow [*1*].
AMPS		Current flow [*1*]. Use **FORCE** for type.
CHRG		Charge [*1*]. Use **FORCE** for type.
FLUX		Magnetic flux [*1*].
CSG	X, Y, Z	X, Y, or Z magnetic current segment component [*1*].
RATE		Diffusion flow rate
SENE		"Stiffness" energy or thermal heat dissipation. Same as TENE [*1*].
STEN		Elemental energy dissipation due to stabilization [*1*].
TENE		Thermal heat dissipation or "stiffness" energy. Same as SENE [*1*].

KENE		Kinetic energy [*1*].
AENE		Artificial energy due to hourglass control/drill stiffness or due to contact stabilization [*1*].
JHEAT		Element Joule heat generation [*1*].
JS	X, Y, Z, SUM	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM). [*1*].
JT	X, Y, Z, SUM	Total measureable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM). [*1*].
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [*1*].
MRE		Magnetic Reynolds number [*1*].
VOLU		Volume of volume element [*1*].
CENT	X, Y, Z	Centroid X, Y, or Z location (based on shape function) in the active coordinate system [*1*].
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
SMISC	*snum*	Element summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each element description found in the *Element Reference*).
NMISC	*snum*	Element non-summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each element description found in the *Element Reference*).

CAP	C0,X0,K0, ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; I1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2: shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.
EDPC	CSIG,CSTR	Material EDP creep model only (not including the cap model): Equivalent creep stress; equivalent creep strain.
FICT	TEMP	Fictive temperature.
	X,Y,Z,XY, YZ,ZX	Components of Biot's effective stress.
ESIG	1, 2, 3	Principal stresses of Biot's effective stress.
	INT	Stress intensity of Biot's effective stress.
	EQV	Equivalent stress of Biot's effective stress.
	TPOR	Total porosity (Gurson material model).
DPAR	GPOR	Porosity due to void growth.
	NPOR	Porosity due to void nucleation.
FFLX	X,Y,Z	Fluid flow flux in poromechanics.

1. Not supported by PowerGraphics
2. For MPC-based contact definitions, the value of STAT can be negative. This indicates that one or more contact constraints were intentionally removed to prevent overconstraint. STAT = -3 is used for MPC bonded contact; STAT = -2 is used for MPC no-separation contact.
3. Works only if failure criteria information is provided. (For more information, see the documentation for the **FC** and **TB** commands.)
4. Works only if user-defined failure criteria routine is provided.
5. Must be added via the **FCTYP** command first.
6. Some element- and material-type limitations apply. (For more information, see the documentation for the **PRERR** command.)

7. Do not use the **PLESOL** command to obtain contact forces for contact elements. The force values reported by this command may not be accurate for these elements. Instead, use the **ETABLE** command to obtain contact force values.
8. Total mechanical, thermal, and ddiffusion strain (EPEL + EPPL + EPCR + EPTH + EPDI) in coupled-diffusion analyses.

Menu Paths

Main Menu>General Postproc>Plot Results>Contour Plot>Element Solu

Utility Menu>Plot>Results>Contour Plot>Elem Solution

VCLEAR

VCLEAR, *NV1*, *NV2*, *NINC*
Deletes nodes and volume elements associated with selected volumes.

PREP7: Meshing
MP ME ST PR PRN <> <> FL EM EH DY PP EME MFS

NV1*, *NV2*, *NINC

Delete mesh for volumes *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and mesh for all selected volumes [**VSEL**] is deleted. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

Notes

Deletes *all* nodes and volume elements associated with selected volumes (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed volumes and nodes associated with non-volume elements will not be deleted. Attributes assigned as a result of **VATT** are maintained. In the program's response to the command, if a volume, area, line, or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Volumes