Protel 99 SE

PCB ASCII File Format

Reference

Introduction

The Design Explorer 99 SE's PCB editor provides different file format options when saving Protel PCB documents. The default binary format generates a compact database, which loads and saves PCB data efficiently. PCB documents are also saved in a Protel ASCII text format. In this document, the ASCII PCB file format for Protel PCB 99 SE will only be documented.

The PCB editor uses a 32 bit design database and can generate through hole and SMD designs of up to 32 signal layers, plus 16 mid layer power planes. There are 16 mechanical drawing layers which allow you to generate fabrication and assembly drawings for your design. The boards can be as big as 100 inches by 100 inches. The Placement accuracy on the 0.001 mils grid system is ±0.0005 mils. Prior to the Design Explorer 99SE, there were up to 16 signal layers, 4 mid layer power planes and 4 mechanical drawing layers.

Signal layers

There are 32 signal layers that can be used for track placement. Anything placed on these layers will be plotted as solid (copper) areas on the PCB.

Internal Planes

Sixteen layers (named Internal Plane 1–16) are available for use as power planes. Nets can be assigned to these layers and multi-layer pads and vias automatically connect to these planes.

SilkScreen layers

Top and Bottom Silkscreen layers are typically used to display component outlines and component text (designator and comment fields that are part of the component description).

Mechanical Layers

Sixteen mechanical drawing layers are provided for fabrication and assembly details such as dimensions, alignment targets, annotation or other details. Mechanical layer items can be automatically added to other layers when printing or plotting artwork. Mechanical Layer names are user definable.

Solder Mask

Top and bottom masks are provided for creating the artwork used to make the solder masks.

Paste Masks

Top and bottom masks are provided to generate the artwork which is used to manufacture stencils to deposit solder paste onto surface mount pads on PCBs with surface mount devices (SMDs).

Drill Drawing

Coded plots of board hole locations typically used to create a drilling drawing that shows a unique symbol for each hole size at each hole location. Individual layer pair plots are provided when blind/buried vias are specified. Three symbol styles are available: coded symbol, alphabetical codes (A, B, C etc.) or the assigned size.

Drill Guide

Plots of all holes in the layout - sometimes called *pad masters*. Individual layer pair plots are provided when blind/buried vias are specified. These plots include all pads and vias with holes greater than zero (0) size.

Keep Out

This layer is used to define the regions where components and routes can validly be placed.

Multi Layer

Objects placed on this layer will appear on all copper layers. This is typically used for through hole pads and vias, but other objects can be placed on this layer

Display Layers

The system layers described below cannot have objects placed on them but they are turned on or off in the Design>>Options dialog box.

DRC Errors- This option controls the display of the Design Rule Check (DRC) error marker.

Visible Grids- Controls the display of the two visible grids.

Pad and Via Holes- Controls the display of pad and via holes. To be able to distinguish pads from vias in draft mode, pad holes are outlined in the current Pad Holes colour (set in the Colours Tab of the Preferences dialog box).

Connect- This option controls the display of the connection lines. The PCB editor displays connection lines wherever it locates part of a net that is un-routed.

The structure of a PCB ASCII file

The PCB ASCII file format is self explanatory with all values and parameters named in the file itself. The format is arranged into records with each record taking a line in the file. The records are sorted by record name. The record names are in the order of creation: Board, ECO, Output, Printer, Gerber, AdvancedPlacer, DesignRuleChecker, Classes, Nets, Components, Polygons, Dimensions, Coordinates, Rules, Connections, Embeddeds, Arcs, Pads, Vias, Tracks, Texts, Fills and FromTos. See the table below for a succinct explanation of each section. In the records of a PCB file, you will see defined PCB objects. A PCB object can either be a primitive or a group object. Primitives are basic objects used by the PCB editor. For example, an arc is a primitive. A group object is composed of primitives. For example a polygon is a group object that can be made up of arc and track objects.

Section	Description
Board	Board Options that are set up in Design-Options.
ECO	Engineering Change Order Options (not enabled in P99/P99SE).
Output	Output options.
Printer	Printer options.
Gerber	Gerber output settings.
AdvancedPlacer	Advanced Placer options and settings.
DesignRuleChecker	Design Rule Checker options and settings.
Classes	Define classes to be used by the Design Rules.
Nets	Identifies and describes a net.
Components	Identifies and describes a component object. This group object is made up of track, arc, pad, and text objects.
Polygons	Identifies group objects which are composed of track and arc objects.
Dimensions	Identifies group objects which are composed of track and text objects.
Coordinates	Identifies group objects which are composed of track and text objects.
Connections	A connection is the line that connects from one pin to another pin in the net.
Rules	Defines a rule as part of the Design Rules.
Embeddeds	Invisible primitives (Need Protel API to have access to embedded objects).
Arcs	Identifies and describes arc primitives.
Pads	Identifies and describes pad primitives.
Vias	Identifies and describes via primitives.
Tracks	Identifies and describes track primitives.
Texts	Identifies and describes text primitives.
Fills	Identifies and describes fill primitives.
FromTos	Identifies and describes FromTo primitives.

Notes:

The ASCII format versions supported by the Protel PCB editor are:
- PCB ASCII Version 4.0 (PCB99SE) A PCB file will contain 'KIND=Protel_Advanced_PCB|VERSION=3.00' string.
- PCB ASCII Version 3.0 (PCB3, PCB98 and PCB99) A PCB file will contain 'KIND=Protel_Advanced_PCB|VERSION=3.00' string.
- PCB ASCII Version 2.8 The header of this PCB file contains the 'PCB FILE 6 VERSION 2.80' string.
- Autotrax version The header of this PCB file contains the 'PCB FILE 4' or 'PCB FILE 5' strings.
- DOS 3 PCB version. The header of this PCB file contains the 'DOS 3 PCB' string.

The PCB ASCII file format is arranged into records with each record taking a line in the file and each record has parameters and values for the particular PCB object. Thus the backward compatibility for PCB version 3, 98, 99, 99SE onwards is maintained. The string KIND=Protel_Advanced_PCB|VERSION=3.00 is the same for file format versions 3 and 4, therefore it is possible for you to load PCB99SE files in PCB98 but the information unique to PCB99SE will be lost.

Extra notes:

The major difference between PCB3 and PCB98 is the use of 32 bit code and support for Microsoft NT Operating System for Protel PCB98. The difference between PCB98 and P99 is that P99 uses the new database storage technology. The Protel PCB99 SE editor uses more design rules, more layers and attributes. Therefore the P99SE ASCII files will be larger with the extra information.

Units used in this document

1 inch = 1000 Mils

1 inch = 2.54 cm

kInternalUnits   = 10000; {ths Of a mil}

kMaxCoord   = 99999 * kInternalUnits;

kMinCoord   = 0   * kInternalUnits;

The legal range for coordinates is generally 0 –99999.999mils. The X and Y coordinates of PCB objects are stored as a 'coord' value, that is 10000 units = 1 mils. You need to convert from a 'coord' value to mils and vice versa. There are MilsToCoord and CoordToMils functions (as part of the Protel Application Programming Interface in Protel SDK) that convert from internal units to imperial units and vice versa.

Some of the parameters have 'extended' format for the parametric values in the records. For example, the SignalStimulus rule may have its STOPTIME value set to 60.00n. Thus the value, 60.00n is actually $60 * 1 \times 10^{-9}$. However if the value does not have a suffix, it is taken directly, for example if value = 1.000, then the value is actually 1.000. Please see the list below.

'T' : Value := Value * 1e12;

'G' : Value := Value * 1e9;

'M' : If the string contains meg/MEG then

              Value := Value * 1e6

    Otherwise Value := Value * 1e-3

'K' : Value := Value * 1e3;

'U' : Value := Value * 1e-6;

'N' : Value := Value * 1e-9;

'P' : Value := Value * 1e-12;

'F' : Value := Value * 1e-15;

Defaults Section

Board Object

The board object stores board level settings such as the printer options, autoplacement settings, grid settings, and so on. This object represents a PCB document.

Board attributes

FILENAME (string) The backup filename of the current PCB document.

KIND (string) Normally Kind = Protel_Advanced_PCB.

VERSION (string) Normally Version = 3.

DATE (string) the format is day – month – year. Eg. 27-Jul-2000.

TIME (string) the format is Hour:Minutes:Seconds. Eg 15:30:23 (24 hour format).

ORIGINX (mils) Denotes the X axis of the origin of the PCB document.

ORIGINY (mils) Denotes the Y axis of the origin of the PCB document.

BIGVISIBLEGRIDSIZE (mils) Denotes the size of the biggest visible grid size in mils.

VISIBLEGRIDSIZE (mils) Denotes the size of the smallest visible grid size in mils.

ELECTRICALGRIDRANGE (mils) Denotes the electrical grid size.

ELECTRICALGRIDENABLED (True, False) Denotes whether the electrical grid is enabled or not.

SNAPGRIDSIZE (mils) Denotes the snap grid size (X and Y values are set the same).

SNAPGRIDSIZEX (mils) Denotes the snap grid X size (horizontal value set),

SNAPGRIDSIZEY (mils) Denotes the snap grid Y size (vertical value set).

TRACKGRIDSIZE (mils) Denotes the track grid size.

VIAGRIDSIZE (mils) Denotes the via grid size.

COMPONENTGRIDSIZE (mils) The ComponentGridSize is set in mils. (X and Y values are set the same).

COMPONENTGRIDSIZEX (mils) Denotes the component grid X size.

COMPONENTGRIDSIZEY (mils) Denotes the component grid Y size.

CURRENTWORKINGLAYER (string) The current working layer.

LAYER1Name .. Layer82Name (string) Denotes the layer name for the layer X. (X = 1..82).

LAYER1Prev..LAYER82Prev (string) Denotes the layer before the layer X. (X = 1..82).

LAYER1Next..LAYER82Next Denotes the layer after the layer X. (X = 1..82).

LAYER1MechEnabled..LAYER82MechEnabled (True,False) Denotes the mechanical layer at layer X is enabled or not. (X = 1..82).

LAYER1COPTTHICK..LAYER82COPTTHICK (mils) Typically 1.4 mils for the layer X.

LAYER1DIELTYPE..LAYER82DIELTYPE (integer) Typically 0 for the layer X. 0 = NoDielectric, 1 = Core, 2 =PrePreg,3 = SurfaceMaterial.

LAYER1DIELCONST..LAYER82DIELCONST (real) The dielectric constant for the layer. Typically 4.800 units for the layer X.

LAYER1DIELHEIGHT..LAYER82DIELHEIGHT (mils) The height of the dielectric material. Typically 12.6 mils for the layer X.

LAYER1DIELMATERIAL..LAYER82DIELMATERIAL (string) The material for the dielectric layer. Typically FR-4 for the layer X.

LAYERPAIRXLOW =TOP The first layer of the layer pair on layer X (where X = 1..82).

LAYERPAIRXHIGH =BOTTOM The last layer of the layer pair on layer X (where X = 1..82)

LAYERPAIRXDRILLGUIDE (True,False) The Layer pair on layer X will have drill guides or not. (where X = 1 ..82)

LAYERPAIRXDRILLDRAWING (True,False) The Layer pair on layer X will have drill drawings or not.(where X = 1 ..82)

TOGGLELAYERS (a string of 82 boolean values) Toggle the visibility of all layers – 0 denotes the layer is not visible, 1 denotes the layer is visible. The minimum (first) layer to maximum (last) layer, where min layer is the TopLayer, and max layer is the ViaHoleLayer.

PLACEMARKERX1..PLACEMARKERX10 (mils) Denotes a place marker at location X (X = 1..10). There are 10 place markers.

PLACEMARKERY1..PLACEMARKERY10 (mils) Denotes a place marker at location Y (Y= 1..10). There are 10 place markers.

Typical record for the board object.

```
|RECORD=Board|FILENAME=H:\Protel Process help files\Backup of BasicPCBObjects P99SE v4
ASCII.PCB|KIND=Protel_Advanced_PCB|VERSION=3.00|DATE=27-Jul-2000|TIME=15:18:30|ORIGINX=
0.00000000000000E+0000mil|ORIGINY=
0.00000000000000E+0000mil|BIGVISIBLEGRIDSIZE=10000000.000|VISIBLEGRIDSIZE=200000.000|ELECTRICALGRI
DRANGE=
8.00000000000000E+0000mil|ELECTRICALGRIDENABLED=TRUE|SNAPGRIDSIZE=200000.000000|SNAPGRIDSIZEX=2000
00.000000|SNAPGRIDSIZEY=200000.000000|TRACKGRIDSIZE=200000.000000|VIAGRIDSIZE=200000.000000|COMPON
ENTGRIDSIZE=200000.000000|COMPONENTGRIDSIZEX=200000.000000|COMPONENTGRIDSIZEY=200000.000000|CURREN
TWORKINGLAYER=TOP|DOTGRID=FALSE|DISPLAYUNIT=1|PLANE1NETNAME=(No Net)|PLANE2NETNAME=(No
Net)|PLANE3NETNAME=(No Net)|PLANE4NETNAME=(No Net)|PLANE5NETNAME=(No Net)|PLANE6NETNAME=(No
Net)|PLANE7NETNAME=(No Net)|PLANE8NETNAME=(No Net)|PLANE9NETNAME=(No Net)|PLANE10NETNAME=(No
Net)|PLANE11NETNAME=(No Net)|PLANE12NETNAME=(No Net)|PLANE13NETNAME=(No Net)|PLANE14NETNAME=(No
Net)|PLANE15NETNAME=(No Net)|PLANE16NETNAME=(No Net)
|RECORD=Board|TOPTYPE=3|TOPCONST=3.500|TOPHEIGHT= 4.00000000000000E-0001mil|TOPMATERIAL=Solder
Resist|BOTTOMTYPE=3|BOTTOMCONST=3.500|BOTTOMHEIGHT= 4.00000000000000E-
0001mil|BOTTOMMATERIAL=Solder
Resist|LAYERSTACKSTYLE=0|SHOWTOPDIELECTRIC=FALSE|SHOWBOTTOMDIELECTRIC=FALSE|LAYER1NAME=TopLayer|LA
YER1PREV=0|LAYER1NEXT=39|LAYER1MECHENABLED=FALSE|LAYER1COPTHICK=
1.40000000000000E+0000mil|LAYER1DIELTYPE=1|LAYER1DIELCONST=4.800|LAYER1DIELHEIGHT=
1.26000000000000E+0001mil|LAYER1DIELMATERIAL=FR-
4|LAYER2NAME=MidLayer1|LAYER2PREV=0|LAYER2NEXT=0|LAYER2MECHENABLED=FALSE|LAYER2COPTHICK=
1.40000000000000E+0000mil|LAYER2DIELTYPE=0|LAYER2DIELCONST=4.800|LAYER2DIELHEIGHT=
1.26000000000000E+0001mil|LAYER2DIELMATERIAL=FR-
4|..|LAYERPAIR0LOW=TOP|LAYERPAIR0HIGH=BOTTOM|LAYERPAIR0DRILLGUIDE=FALSE|LAYERPAIR0DRILLDRAWING=FAL
SE
|RECORD=Board|TOGGLELAYERS=10000000000000000000000000000001100001000000000000000011000000000000000
00110100111|PLACEMARKERX1=-1.00000000000000E-0004mil|PLACEMARKERY1=-1.00000000000000E-
0004mil|PLACEMARKERX2=-1.00000000000000E-0004mil|PLACEMARKERY2=-1.00000000000000E-
0004mil|PLACEMARKERX3=-1.00000000000000E-0004mil|PLACEMARKERY3=-1.00000000000000E-
0004mil|PLACEMARKERX4=-1.00000000000000E-0004mil|PLACEMARKERY4=-1.00000000000000E-
0004mil|PLACEMARKERX5=-1.00000000000000E-0004mil|PLACEMARKERY5=-1.00000000000000E-
0004mil|PLACEMARKERX6=-1.00000000000000E-0004mil|PLACEMARKERY6=-1.00000000000000E-
0004mil|PLACEMARKERX7=-1.00000000000000E-0004mil|PLACEMARKERY7=-1.00000000000000E-
0004mil|PLACEMARKERX8=-1.00000000000000E-0004mil|PLACEMARKERY8=-1.00000000000000E-
0004mil|PLACEMARKERX9=-1.00000000000000E-0004mil|PLACEMARKERY9=-1.00000000000000E-
0004mil|PLACEMARKERX10=-1.00000000000000E-0004mil|PLACEMARKERY10=-1.00000000000000E-0004mil
```

Output Options

The output options defines the attributes such as including un-connected plots, specifying the drill symbol kind for plotting PCB documents.

Output attributes

DRILLGUIDEHOLESIZE (Mils) Denotes the size of the drill guide hole.

DRILLDRAWSYMBOLSIZE (Mils) Denotes the size of the drill draw symbol.

DRILLSYMBOLKIND (Integer) Denotes the drill symbol kind.

MULTILAYERONPADMASTER (True,False). Denotes that the multi layer is to be used or not for the pad master.

TOPLAYERONPADMASTER (True,False) Denotes that the top layer is to be used or not for the pad master.

BOTTOMLAYERONPADMASTER (True,False) Denotes that the bottom layer is to be used or not for the pad master.

INCLUDEVIASINSOLDERMASK (True,False) Denotes that vias are to be included in the solder mask.

INCLUDEMECHXWITHALLPLOTS (True,False) Denotes whether or not mechanical layer X is to be included with all plots (X = 1..16)

INCLUDEUNCONNECTEDPADS (True,False) Denotes whether or not unconnected pads are to be included with all plots.

PLOTLAYERS (a string of 109 boolean values) A string of plot layers that are to be plotted. A value of 1 denotes this layer will be plotted, and the value of 0 denotes that this layer will not be plotted. This is represented as an array of eTopLayerPlot to eBottomPadmasterPlot.

FLIPLAYERS (a string of 109 boolean values) A string of plot layers that are to be mirrored. A value of 1 denotes this layer will be plotted, and the value of 0 denotes that this layer will not be plotted. This is represented as an array of eTopLayerPlot to eBottomPadmasterPlot.

Typical record for the Output Options

```
|RECORD=OutputOptions|DRILLGUIDEHOLESIZE= 3.00000000000000E+0001mil|DRILLDRAWSYMBOLSIZE=
5.00000000000000E+0001mil|DRILLSYMBOLKIND=24|MULTILAYERONPADMASTER=TRUE|TOPLAYERONPADMASTER=TRUE|B
OTTOMLAYERONPADMASTER=TRUE|INCLUDEVIASINSOLDERMASK=TRUE|INCLUDEUNCONNECTEDPADS=TRUE|INCLUDEMECH1WI
THALLPLOTS=FALSE|INCLUDEMECH2WITHALLPLOTS=FALSE|INCLUDEMECH3WITHALLPLOTS=FALSE|INCLUDEMECH4WITHALL
PLOTS=FALSE|INCLUDEMECH5WITHALLPLOTS=FALSE|INCLUDEMECH6WITHALLPLOTS=FALSE|INCLUDEMECH7WITHALLPLOTS
=FALSE|INCLUDEMECH8WITHALLPLOTS=FALSE|INCLUDEMECH9WITHALLPLOTS=FALSE|INCLUDEMECH10WITHALLPLOTS=FAL
SE|INCLUDEMECH11WITHALLPLOTS=FALSE|INCLUDEMECH12WITHALLPLOTS=FALSE|INCLUDEMECH13WITHALLPLOTS=FALSE
|INCLUDEMECH14WITHALLPLOTS=FALSE|INCLUDEMECH15WITHALLPLOTS=FALSE|INCLUDEMECH16WITHALLPLOTS=FALSE
|RECORD=OutputOptions|PLOTLAYERS=000000000000000000000000000000000000000000000000000000000000000
0000000000000000000000000000000000000000000|FLIPLAYERS=0000000000000000000000000000000000000000000
00000000000000000000000000000000000000000000000000000000000000000
```

Printer Options

The printer options defines the attributes for a printer, such as the size and offsets of printed pages.

Printer attributes

DEVICE (string) Denotes the name of the device. Normally the Design Explorer requests the printing services from the Operating System.

DRIVER (string) Denotes the name of the driver. Normally the Design Explorer requests the printing services from the Operating System.

OUTPUT (string) Denotes the name of the output. Normally the Design Explorer requests the printing services from the Operating System.

SHOWHOLES (True,False) Denotes whether or not to display holes of pads.

SCALETOFIT (True, False) Denotes whether or not the output is scaled to fit the page.

USEPRINTERFONTS (True,False) Denotes whether printer fonts are used or not.

USESOFTWAREARCS (True,False) Denotes whether software arcs are used or not.

BATCHTYPE (0..1) 0 = PlotPerSheet, 1 = Panelize.

COMPOSITETYPE (0..1). 0 = color, 1 = mono.

cBORDERSIZE (Mils) Normally the border size is 1000 mils.

SCALE (Mils) Denotes the scaling factor of the output. A one to one relationship is a scale factor of 1.

XCORRECT (mils) Denotes the X axis correction factor.

YCORRECT (mils) Denotes the Y axis correction factor.

PLOTPADNETS (True,False) Denotes whether to plot pad net information or not.

PLOTPADNUMBERS (True,False) Denotes whether to plot the pad numbers or not.

PLOTTERSCALE (mils) Denotes the scale factor for the plotter.

PLOTTERXCORRECT (mils) Denotes the plotter X axis offset.

PLOTTERYCORRECT (mils) Denotes the plotter Y axis offset.

PLOTTERXOFFSET (mils) Denotes the X axis offset.

PLOTTERYOFFSET (mils) Denotes the Y axis offset.

PLOTTERSHOWHOLES (True, False) Denotes whether the plotter will show holes or not.

PLOTTERUSEOFARCS (True, False) Denote s whether the plotter will use arcs or not.

PLOTTERWAITBETWEENSHEETS (True, False) Denotes whether the plotter will wait between paper sheets or not.

PLOTTEROUTPUTPORT (0..4) The port value (0..4) corresponds to one of the following (Com1, Com2, Com3, Com4, File) values.

PLOTTERLANGUAGE (0..1) Denotes what language (0 = HPGL, 1 = DMPL) the plotter is using.

*SPD*1..8 (integer) Denotes the speed of the plotter. Normally 3.

*THK*1..8 (mils) Denotes the thickness of the plotter pens. Normally 1.3 mils.

PLOTMODE (0..2) The plot mode value (0..2) corresponds to one of the following (DrawFull, DrawDraft, DrawHidden) values for each layer (eArcObject to eEmbeddedObject).

DRIVERTYPE (0..5) Denotes the driver type which corresponds to one of the following (UnknownDriver, ProtelGerber, ProtelPlot_Composite, ProtelPlot_Final, StandardDriver_Composite, StandardDriver_Final) values.

PLOTLAYERS (a string of 82 boolean values) A string of plot layers that are to be plotted. A value of 1 denotes this layer will be plotted, and the value of 0 denotes that this layer will not be plotted. This is represented as an array of TopLayerPlot layer to BottomPadMasterPlot layer.

*PP*1..82 (integer) Type of plot pen for each layer.

*PM*1..82 (integer) Color value a RGB value of TColor type for each monochrome plot layer.

*PC*1..82 (integer) Color value a RGB value of TColor type for each color plot layer.

Notes:

Color = RedVal + 256*(GreenVal + 256*BlueVal). Where RedVal is an integer expression from 0 to 255 representing the red level in an RGB color specification. GreenVal is an integer expression from 0 to 255 representing the green level in an RGB color specification. BlueVal is an integer expression from 0 to 255 representing the blue level in an RGB color specification.

Typical record for Printer Options

```
|RECORD=PrinterOptions|DEVICE=|DRIVER=|OUTPUT=|SHOWHOLES=FALSE|SCALETOFITPAGE=FALSE|USEPRINTERFONT
S=FALSE|USESOFTWAREARCS=FALSE|BATCHTYPE=0|COMPOSITETYPE=0|CBORDERSIZE=
1.00000000000000E+0003mil|SCALE=0.0001mil|XCORRECT=0.0001mil|YCORRECT=0.0001mil
|RECORD=PrinterOptions|PLOTPADNETS=TRUE|PLOTPADNUMBERS=TRUE|PLOTTERSCALE=0.0001mil|PLOTTERXCORRECT
=0.0001mil|PLOTTERYCORRECT=0.0001mil|PLOTTERXOFFSET= 0.00000000000000E+0000mil|PLOTTERYOFFSET=
0.00000000000000E+0000mil|PLOTTERSHOWHOLES=FALSE|PLOTTERUSESOFTWAREARCS=FALSE|PLOTTERWAITBETWEENSH
EETS=FALSE|PLOTTEROUTPUTPORT=0|PLOTTERLANGUAGE=0
|RECORD=PrinterOptions|SPD1=3|THK1= 1.30000000000000E+0001mil|SPD2=3|THK2=
1.30000000000000E+0001mil|SPD3=3|THK3= 1.30000000000000E+0001mil|SPD4=3|THK4=
1.30000000000000E+0001mil|SPD5=3|THK5= 1.30000000000000E+0001mil|SPD6=3|THK6=
1.30000000000000E+0001mil|SPD7=3|THK7= 1.30000000000000E+0001mil|SPD8=3|THK8=
1.30000000000000E+0001mil
|RECORD=PrinterOptions|PLOTMODE=00000000000000000|DRIVERTYPE=0
|RECORD=PrinterOptions|PLOTLAYERS=1111111111111111111111111111111111111111111111111111111111111111
11111111111111111111
|RECORD=PrinterOptions|PP1=1|PP2=1|PP3=1|PP4=1|PP5=1|PP6=1|PP7=1|PP8=1|PP9=1|PP10=1|PP11=1|PP12=1|
PP13=1|PP14=1|PP15=1|PP16=1|PP17=1|PP18=1|PP19=1|PP20=1|PP21=1|PP22=1|PP23=1|PP24=1|PP25=1|PP26=1|
PP27=1|PP28=1|PP29=1|PP30=1|PP31=1|PP32=1|PP33=1|PP34=1|PP35=1|PP36=1|PP37=1|PP38=1|PP39=1|PP40=1|
PP41=1|PP42=1|PP43=1|PP44=1|PP45=1|PP46=1|PP47=1|PP48=1|PP49=1|PP50=1|PP51=1|PP52=1|PP53=1|PP54=1|
PP55=1|PP56=1|PP57=1|PP58=1|PP59=1|PP60=1|PP61=1|PP62=1|PP63=1|PP64=1|PP65=1|PP66=1|PP67=1|PP68=1|
PP69=1|PP70=1|PP71=1|PP72=1|PP73=1|PP74=1|PP75=1|PP76=1|PP77=1|PP78=1|PP79=1|PP80=1|PP81=1|PP82=1
|RECORD=PrinterOptions|PM1=2105376|PM2=394758|PM3=789516|PM4=1184274|PM5=1579032|PM6=1973790|PM7=2
368548|PM8=2763306|PM9=3158064|PM10=3552822
|RECORD=PrinterOptions|PM11=3947580|PM12=4342338|PM13=4737096|PM14=5131854|PM15=5526612|PM16=59213
70|PM17=6316128|PM18=6710886|PM19=7105644|PM20=7500402
|RECORD=PrinterOptions|PM21=7895160|PM22=8289918|PM23=8684676|PM24=9079434|PM25=9474192|PM26=98689
50|PM27=10263708|PM28=10658466|PM29=11053224|PM30=11447982
|RECORD=PrinterOptions|PM31=11842740|PM32=4210752|PM33=12632256|PM34=6710886|PM35=7105644|PM36=750
0402|PM37=7895160|PM38=8289918|PM39=8684676|PM40=9079434
|RECORD=PrinterOptions|PM41=9474192|PM42=9868950|PM43=16579836|PM44=131586|PM45=526344|PM46=921102
|PM47=1315860|PM48=1710618|PM49=2105376|PM50=2500134
|RECORD=PrinterOptions|PM51=2894892|PM52=3289650|PM53=3684408|PM54=4079166|PM55=10263708|PM56=1065
8466|PM57=11053224|PM58=11447982|PM59=11842740|PM60=12237498
|RECORD=PrinterOptions|PM61=6842472|PM62=7237230|PM63=7631988|PM64=8026746|PM65=8421504|PM66=88162
62|PM67=9211020|PM68=9605778|PM69=10000536|PM70=10395294
|RECORD=PrinterOptions|PM71=10790052|PM72=11184810|PM73=12632256|PM74=0|PM75=13421772|PM76=1677721
5|PM77=14211288|PM78=14606046|PM79=15000804|PM80=15395562
|RECORD=PrinterOptions|PM81=15790320|PM82=16185078
|RECORD=PrinterOptions|PC1=255|PC2=128|PC3=32768|PC4=65280|PC5=8388608|PC6=16776960|PC7=8388736|PC
8=16711935|PC9=32896|PC10=65535
|RECORD=PrinterOptions|PC11=8421504|PC12=32768|PC13=8388736|PC14=8421376|PC15=12632256|PC16=128|PC
17=32768|PC18=65280|PC19=8388608|PC20=16776960
|RECORD=PrinterOptions|PC21=8388736|PC22=16711935|PC23=32896|PC24=65535|PC25=8421504|PC26=32768|PC
27=8388736|PC28=8421376|PC29=12632256|PC30=128
|RECORD=PrinterOptions|PC31=32768|PC32=16711680|PC33=32768|PC34=7585984|PC35=8388736|PC36=128|PC37
=3162822|PC38=7307173|PC39=32768|PC40=128
|RECORD=PrinterOptions|PC41=8388736|PC42=8421376|PC43=32768|PC44=128|PC45=8388736|PC46=8421376|PC4
7=32768|PC48=128|PC49=8388736|PC50=8421376
|RECORD=PrinterOptions|PC51=32768|PC52=128|PC53=8388736|PC54=8421376|PC55=128|PC56=8388736|PC57=83
88736|PC58=8421376|PC59=32768|PC60=0
|RECORD=PrinterOptions|PC61=8388736|PC62=8421376|PC63=32768|PC64=0|PC65=8388736|PC66=8421376|PC67=
32768|PC68=0|PC69=8388736|PC70=8421376
|RECORD=PrinterOptions|PC71=32768|PC72=0|PC73=3408013|PC74=8421504|PC75=8421376|PC76=15269887|PC77
=65280|PC78=65535|PC79=12632256|PC80=11913679
|RECORD=PrinterOptions|PC81=6899487|PC82=9279142
```

Gerber Options

The photoplotters are controlled by a vector-based plotting language, Gerber language, -- a registered trademark of the Gerber Scientific Company. This language has become an industry standard (also known as RS-274). A Gerber format file describes a plot as a series of draft codes (or commands) and coordinates. The gerber output options are controlled by the gerber settings in the PCB editor.

Gerber attributes

SORTOUTPUT (True,False) Denotes whether to sort the vector photoplot or not.

USESOFTWAREARCS (True,False) Denotes whether to use software arcs for the vector photoplot.

CENTERPHOTOPLOTS (True,False) Denotes whether to center the photoplots.

EMBEDAPERTURES (True,False) Denotes whether to embed the apertures in the photoplot file.

PANELIZE (True,False) Denotes whether to panelize the output or not.

G54 (True,False) Denotes whether to use the G54 command code when photoplotting. On some older plotters the special code "G54" needs to be sent before each change of aperture code.

PLUSTOL (True,False) Denotes the match tolerance.

MINUSTOL (True,False) Denotes the match tolerance.

FILMSIZEX (True,False) Sets the file size in X coordinates for the target photoplotter.

FILMSIZEY (True,False) Sets the file size in Y coordinates for the target photoplotter.

BORDERSIZE (True,False) Sets the border size of the film for the target photoplotter.

APTTABLE (True,False) Denotes whether or not the aperture file will be loaded or not. Normally unused.

MAXAPERSIZE (True,False) Denotes the maximum aperture size.

RELIEFSHAPESALLOWED (True,False) Denotes whether or not the relief shapes are allowed.

PADSFLASHONLY (True,False) Denotes whether or not pads are to be flashed only.

GERBERUNITS (Integer) Denotes the gerber units. Please refer to the table below.

GERBERDECS (Integer) Denotes a range of 3 to 5 decimal points.

GerberDecs and GerberUnits relationship:
GerberDecs =3 : GerberUnits = 1
GerberDecs =4 : GerberUnits = 10
GerberDecs =5 : GerberUnits = 100

Typical record for the Gerber Options.

```
|RECORD=GerberOptions|SORTOUTPUT=FALSE|USESOFTWAREARCS=TRUE|CENTERPHOTOPLOTS=TRUE|EMBEDAPERTURES=F
ALSE|PANELIZE=FALSE|G54=FALSE|PLUSTOL= 5.00000000000000E-0003mil|MINUSTOL= 5.00000000000000E-
0003mil|FILMSIZEX= 2.00000000000000E+0004mil|FILMSIZEY= 1.60000000000000E+0004mil|BORDERSIZE=
1.00000000000000E+0003mil|APTTABLE=|MAXAPERSIZE=
2.50000000000000E+0002mil|RELIEFSHAPESALLOWED=TRUE|PADSFLASHONLY=TRUE|GERBERUNITS=1|GERBERDECS=1
```

Advanced Placer Options

The Advanced Placer is a server that places components on a PCB document according to the placement options set by a user.

Advanced Placer attributes

PLACELARGECLEAR (Mils) Denotes the large component clearance.

PLACESMALLCLEAR (Mils) Denotes the small component clearance.

PLACEUSEROTATION (True, False) Denotes whether components will be rotated (in increments of 90 degrees) or not during auto placement.

PLACEUSELAYERSWAP (True, False) Denotes whether layers will be swapped or not during auto placement.

PLACEBYPASSNET1 (string) If a first net is specified, then the components will be automatically placed with their associated integrated circuits. This net can be assigned to indicate By-Pass capacitors (commonly VCC and GND).

PLACEBYPASSNET2 (string) If a second net is specified, then the components will be automatically placed with their associated integrated circuits. This net can be assigned to indicate By-Pass capacitors (commonly VCC and GND).

PLACEUSEADVANCEDPLACE (True, False) Denotes whether component placement will be done or not.

PLACEUSEGROUPING (True, False) When the PlaceUseGrouping is true, the system looks for component groupings based on the connections present in the netlist.

Typical record for Advanced Placer Options

```
|RECORD=AdvancedPlacerOptions|PLACELARGECLEAR= 5.00000000000000E+0001mil|PLACESMALLCLEAR=
2.00000000000000E+0001mil|PLACEUSEROTATION=TRUE|PLACEUSELAYERSWAP=FALSE|PLACEBYPASSNET1=|PLACEBYPA
SSNET2=|PLACEUSEADVANCEDPLACE=TRUE|PLACEUSEGROUPING=TRUE
```

Design Rule Checker Options

The Design Rule Checker (DRC) checks the status of design rules specifications and control in the PCB editor.

Design Rule Checker system attributes

DOMAKEDRCFILE (True, False) If this is true, a DRC report file will be created.

DOMAKEDRCERRORLIST (True, False) If this is true, a DRC report will contain a list of errors.

DOSUBNETDETAILS (True, False) If this is true, and if unrouted net rule is defined, then the sub net details will be included in the DRC report.

RULESETTOCHECK(an array of defined design rules) Denotes which 49 rules to check.

ONLINERULESETTOCHECK (an array of defined design rules.) Denotes which 49 rules to check online.

REPORTFILENAME (string) Denotes the report file name. Default is blank.

EXTERNALNETLISTFILENAME (string) Denotes the file name of an external net list.

CHECKEXTERNALNETLIST (True, False) If true, the external net list is checked against the DRC system.

MAXVIOLATIONCOUNT (Integer) The DRC will stop after logging the MaxViolationCount violations. The default value is 30.

INTERNALPLANEWARNINGS (True, False) If this is true, internal plane warnings will be logged in the DRC report.

Typical record for Design Rule Checker options

```
|RECORD=DesignRuleCheckerOptions|DOMAKEDRCFILE=TRUE|DOMAKEDRCERRORLIST=TRUE|DOSUBNETDETAILS=TRUE|R
EPORTFILENAME=|EXTERNALNETLISTFILENAME=|CHECKEXTERNALNETLIST=FALSE|MAXVIOLATIONCOUNT=500|RULESETTO
CHECK=0,1,2,3,4,5,15,16,18,19,21,22,23,42,45,46,47|ONLINERULESETTOCHECK=0,1,22,23,24,48|INTERNALPL
ANEWARNINGS=TRUE
```

Classes

A class is defined as a group or set of objects identified by its unique class name. The PCB Editor allows you to create classes (sets) of objects which are very handy for setting up and managing the design rules. The PCB editor supports Net classes, Component classes, From-To classes and Pad classes.

Class attributes

NAME (string) There are four system classes: All components, All Nets, All From-Tos, All Pads, or user defined class.

KIND (integer) If the Kind value = 2, it denotes pads, 8 denotes nets, 9 denotes components, 15 denotes FromTo objects.

SUPERCLASS (True,False) If true, all members of the super class will be identified. If false, the current class will be identified.

SELECTED (True,False) Denotes whether the set of objects identified by the class will be selected or not.

Typical record for classes

```
|RECORD=Class|NAME=All Components|KIND=9|SUPERCLASS=TRUE|SELECTED=FALSE
|RECORD=Class|NAME=All Nets|KIND=8|SUPERCLASS=TRUE|SELECTED=FALSE
|RECORD=Class|NAME=All From-Tos|KIND=15|SUPERCLASS=TRUE|SELECTED=FALSE
|RECORD=Class|NAME=All Pads|KIND=2|SUPERCLASS=TRUE|SELECTED=FALSE
```

PCB Objects Section

Net objects

A net contains information about the components used in the design, and the connectivity created in the design in the form of nets. A net is a list of component pins that are electrically connected in the design. The pin to pin connections are displayed for each net. The arrangement of the pin to pin connections is called the net topology.

Net Attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

NAME (String) Denotes the name of a net object.

VISIBLE (True / False) Denotes the visibility of the pin to pin connections. If true, the net is not displayed in the ratsnest. If false, the connections lines for the net will be shown.

COLOR (Integer) The color = RedVal + 256*(GreenVal + 256*BlueVal). Where RedVal is an integer expression from 0 to 255 representing the red level in an RGB color specification. GreenVal is an integer expression from 0 to 255 representing the green level in an RGB color specification. BlueVal is an integer expression from 0 to 255 representing the blue level in an RGB color specification.

Typical Net object record

```
|RECORD=Net|ID=0|SELECTION=FALSE|NAME=+12|VISIBLE=TRUE|COLOR=8421376
```

Component Objects

Components are any collection of primitives (tracks, pads, arcs, etc) that have been selected as a group and then added to the component library. Both through-hole and SMD components can be placed on either side (Top layer or Bottom layer) of the board. A component has comment and designator fields. These fields can be placed anywhere relative to the component and can be placed on copper or overlay layer.

Component Attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a component is on.

LOCKED (True, False) Denotes whether a component object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if a component object is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether a component object has been routed manually or not.

PRIMITIVELOCK (True,False) If true, the component is protected from being edited graphically. Lock a component whose position is critical. If you try to edit a primitive that is locked, you will be informed that the primitive is locked and asked if you wish to proceed with the action. If false, the primitive can be freely edited without confirmation

X (Mils) Denotes the X location of a component object.

Y (Mils) Denotes the Y location of a component object.

PATTERN (String). The footprint of a component is also called a pattern.

NAMEON (True,False) Denotes the visibility of a Name text object.

COMMENTON (True,False) Denotes the visibility of a Comment text object.

GROUPNUM (Integer) Not used.

COUNT (Integer) The number of primitives in a component.

ROTATION (0 – 360 degrees of real type) Denotes the rotation of a component in degrees.

HEIGHT (Mils) Denotes the height of a component object. This field is used for 3D viewer and other applications which require heights of components to be displayed in three dimensions.

NAMEAUTOPOSITION (0..9) The name text object is to be positioned relative to the component object. The value, 0 denotes eAutoPos_Manual, 1 = eAutoPos_TopLeft, 2 = eAutoPos_CenterLeft, 3 = eAutoPos_BottomLeft, 4= eAutoPos_TopCenter, 5 = eAutoPos_CenterCenter, 6 =eAutoPos_BottomCenter, 7= eAutoPos_TopRight, 8= eAutoPos_CenterRight, 9= eAutoPos_BottomRight.

COMMENTAUTOPOSITION (0..9) The comment text object is to be positioned relative to the component object. The value, 0 denotes eAutoPos_Manual, 1 = eAutoPos_TopLeft, 2 = eAutoPos_CenterLeft, 3 = eAutoPos_BottomLeft, 4= eAutoPos_TopCenter, 5 = eAutoPos_CenterCenter, 6 =eAutoPos_BottomCenter, 7= eAutoPos_TopRight, 8= eAutoPos_CenterRight, 9= eAutoPos_BottomRight.

UNIONINDEX (Integer) Denotes the union index. Unions are sets of components that will be manipulated as a block for the PCB placement process. Components in a union maintain their relative positions within the union as they are moved.

Typical Component object record

```
|RECORD=Component|ID=0|SELECTION=FALSE|LAYER=TOP|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE
|PRIMITIVELOCK=TRUE|X= 2.36000000000000E+0003mil|Y=
1.25600000000000E+0004mil|PATTERN=0402|NAMEON=TRUE|COMMENTON=TRUE|GROUPNUM=0|COUNT=0|ROTATION=0.00
0|HEIGHT= 0.00000000000000E+0000mil|NAMEAUTOPOSITION=0|COMMENTAUTOPOSITION=0|UNIONINDEX=0
```

Polygon Objects

Polygon planes (copper pours) are areas of copper formed when the Polygon plane process is executed. Polygons can be poured to created a multi sided shape on any layer and pour around existing objects, in accordance with the design rules. Polygons consist of tracks and arcs.

Polygon attributes

ID (integer) Normally 0. Not used.

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a polygon is on.

LOCKED (True, False) Denotes whether an arc object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if a polygon is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether a polygon has been routed manually or not

PRIMITIVELOCK (True,False) If true, the polygon is protected from being edited graphically. Lock a polygon whose position is critical. If you try to edit a primitive that is locked, you will be informed that the primitive is locked and asked if you wish to proceed with the action. If false, the primitive can be freely edited without confirmation

POLYGONTYPE (Polygon,Split Plane). Denotes the type of polygon, as a plain polygon or as a split plane.

POUROVER (True,False) Denotes a polygon is to be repoured if true, especially when polygons have been modified.

REMOVEDEAD (True,False) If true, the dead copper is removed.

GRIDSIZE (mils) denoes the grid which the tracks within a polygon are placed. Ideally this grid is a fraction of the component pin pitch, to allow the most effective placement of the polygon tracks.

TRACKWIDTH (mils) Denotes the width of tracks which are placed to form a polygon.

HATCHSTYLE (90Degree,45Degree, VerticalHatch,HorizontalHatch, NoHatching) With the **90 Degree Hatch** value, the polygon is filled with tracks running both horizontally and vertically. This is the default value. With the **45 Degree Hatch** value, the polygon is filled with tracks running at 45 degrees (in both directions). With the **Vertical Hatch** value, the polygon is filled with tracks running vertically. With the **Horizontal Hatch** value the polygon is filled with tracks running horizontally. **No Hatching** the polygon is not filled at all. Only the boundary tracks will be present.

USEOCTAGONS (True,False) Denotes that the octagons are to surround pads if true. If false, pads are surrounded by arcs. Octagons give smaller Gerber files and faster photoplotting.

MINPRIMLENGTH (mils) Denotes the minimum track length used in a polygon.

KIND0..n (Integer) Denotes the kind of segment. 0 = track (line segment) and 1 = arc (circular segment).

VX0..n (mils) Denotes the Vx (the vertex point) of a line segment

VY0..n (mils) Denotes the Vy (the vertex point) of a line segment

CX0..n (mils) Denotes the Cx (the centre) of an arc segment.

CY0..n (mils) Denotes the Cy (the centre) of an arc segment.

SA0..n (mils) Denotes the starting angle of an arc segment

EA0..n (mils) Denotes the ending angle of an arc segment.

R0..n (mils) Denotes the radius of an arc segment.

Notes:

A polygon consists of segments and each segment can be a line or an arc. For a track (line), the kind is a PolySegmentLine which consists of Vx, Vy and Vx+1 and Vy+1 points. For an arc (circular segment), the kind is a PolySegmentArc which consists of Cx,Cy (the center of an arc segment), radius, starting and ending angles.

Typical polygon object record

```
|RECORD=Polygon|ID=0|SELECTION=FALSE|LAYER=TOP|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE|P
RIMITIVELOCK=TRUE|POLYGONTYPE=Polygon|POUROVER=FALSE|REMOVEDEAD=FALSE|GRIDSIZE=
2.00000000000000E+0001mil|TRACKWIDTH=
8.00000000000000E+0000mil|HATCHSTYLE=90Degree|USEOCTAGONS=FALSE|MINPRIMLENGTH=
2.00000000000000E+0001mil|KIND0=0|VX0= 3.02000000000000E+0003mil|VY0=
1.20400000000000E+0004mil|CX0= 0.00000000000000E+0000mil|CY0=
0.00000000000000E+0000mil|SA0=0.000|EA0=0.000|R0= 0.00000000000000E+0000mil|KIND1=0|VX1=
3.16000000000000E+0003mil|VY1= 1.20400000000000E+0004mil|CX1= 0.00000000000000E+0000mil|CY1=
0.00000000000000E+0000mil|SA1=0.000|EA1=0.000|R1= 0.00000000000000E+0000mil|KIND2=0|VX2=
3.16000000000000E+0003mil|VY2= 1.18200000000000E+0004mil|CX2= 0.00000000000000E+0000mil|CY2=
0.00000000000000E+0000mil|SA2=0.000|EA2=0.000|R2= 0.00000000000000E+0000mil|KIND3=0|VX3=
3.02000000000000E+0003mil|VY3= 1.20400000000000E+0004mil|CX3= 0.00000000000000E+0000mil|CY3=
0.00000000000000E+0000mil|SA3=0.000|EA3=0.000|R3= 0.00000000000000E+0000mil
```

Dimension Objects

Dimensions are special entities consisting of text and track primitives. They are automatically generated when starting and ending points are specified after executing the PlaceDimension process.

Dimension attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a dimension object is on.

LOCKED (True, False) Denotes whether a dimension object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if a dimension is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether a dimension has been routed manually or not .

PRIMITIVELOCK (True,False) If true, the dimension is protected from being edited graphically. Lock a dimension whose position is critical. If you try to edit a primitive that is locked, you will be informed that the primitive is locked and asked if you wish to proceed with the action. If false, the primitive can be freely edited without confirmation

X (Mils) Denotes the X location of a dimension object.

Y (Mils) Denotes the Y location of a dimension object.

SIZE (Mils)Denotes the current diameter / side length of the circle / square that encapsulates the leader dimensions' text. Whether the value reflects the diameter for a circle or the length of side for a square, depends on the value set in the Shape field. The default value is 500mil.

LINEWIDTH (Mils) Denotes the current line width used to draw the dimension lines. The default value is 10mil.

TEXTHEIGHT (Mils) Denotes the current text height used to display the dimension values. The default is 60mils. Range is 0.01mils to 10000mils.

TEXTWIDTH (Mils) Denotes the current dimension text stroke width, which defines the "thickness" of the lines used to produce the lettering. The default is 6mils. Range is 0.001 to 255 mils.

FONT (DEFAULT, SANS SERIF, SERIF) This parameter sets the current font used for the dimension text.

STYLE (None, Normal, Bracklets) This parameter sets the current setting for the display style of dimension units. **None**: No units are displayed, only the coordinates [eg. 1220, 3400]. **Normal**: Units are displayed with units after each value [eg. 1220mil, 3400mil]. **Brackets**: Units are displayed in brackets at the end of the dimension [eg. 1220, 3400 (mil)].

Typical dimension object record

```
|RECORD=Dimension|SELECTION=FALSE|LAYER=TOP|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE|PRIM
ITIVELOCK=TRUE|X1= 3.42000000000000E+0003mil|Y1= 1.20000000000000E+0004mil|X2=
4.72000000000000E+0003mil|Y2= 1.19800000000000E+0004mil|HEIGHT=
1.00000000000000E+0001mil|LINEWIDTH= 1.00000000000000E+0001mil|TEXTHEIGHT=
6.00000000000000E+0001mil|TEXTWIDTH= 6.00000000000000E+0000mil|FONT=DEFAULT|STYLE=Brackets
```

Coordinate Objects

Coordinate markers are special entities consisting of text and track primitives. They are automatically generated when a position is specified after executing the PlaceCoordinate process.

Coordinate attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a coordinate object is on.

LOCKED (True, False) Denotes whether a coordinate object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if a coordinate object is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether a coordinate object has been routed manually or not.

PRIMITIVELOCK (True,False) If true, the coordinate is protected from being edited graphically. Lock a coordinate whose position is critical. If you try to edit a primitive that is locked, you will be informed that the primitive is locked and asked if you wish to proceed with the action. If false, the primitive can be freely edited without confirmation

X (Mils) Denotes the X location of a coordinate object.

Y (Mils) Denotes the Y location of a coordinate object.

SIZE (mils) This parameter sets the current length of the marker lines that pinpoint the coordinate's location. The marker lines form a cross centered on the measurement point of the coordinate. The default value is 10mil.

LINEWIDTH (mils) This parameter sets the current line width used to draw the coordinate marker lines, which form a cross centered on the measurement point of the coordinate. The default value is 10mil.

TEXTHEIGHT (mils) This parameter sets the current text height used to display the coordinate text. The default is 60mils. Range is 0.01mils to 10000mils. The character width used to display / print the text is automatically proportioned to the height. A minimum text height of 36 mils will allow strings to be legibly photoplotted.

TEXTWIDTH (mils) This parameter sets the current coordinate text stroke width, which defines the "thickness" of the lines used to produce the lettering. The default is 6mils. Width range is 0.001 to 255 mils. The character width used to display / print the text is automatically proportioned to the height. A minimum text height of 36 mils will allow strings to be legibly photoplotted.

FONT (Default, Sans Serif, Serif) This parameter sets the current font used for the coordinate text.

STYLE (None, Normal, Bracklets) This parameter sets the current setting for the display style of coordinate units. **None**: No units are displayed, only the coordinates [eg. 1220, 3400]. **Normal**: Units are displayed with units after each value [eg. 1220mil, 3400mil]. **Brackets**: Units are displayed in brackets at the end of the coordinate [eg. 1220, 3400 (mil)].

ROTATION (Real value 0 – 360 degrees.) This parameter sets the rotation of a coordinate object.

Typical coordinate object record

```
|RECORD=Coordinate|SELECTION=FALSE|LAYER=TOP|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE|PRI
MITIVELOCK=TRUE|X= 3.74000000000000E+0003mil|Y= 1.17400000000000E+0004mil|SIZE=
1.00000000000000E+0001mil|LINEWIDTH= 1.00000000000000E+0001mil|TEXTHEIGHT=
6.00000000000000E+0001mil|TEXTWIDTH=
6.00000000000000E+0000mil|FONT=DEFAULT|STYLE=Brackets|ROTATION=0.000
```

Rule Objects

The PCB editor incorporates a large set of design rules. The rule records in a PCB file lists all instances of design rules that have been defined for the particular PCB document. The rule scope of each rule defines exactly what objects that rule is to apply to. A compound design rule scope consists of two or more rule scopes "ANDed" together. In other words, for the rule to apply the object must satisfy scope 1 AND scope 2 AND scope 3, etc. The ability to create compound design rule scopes allows you to target almost any set of objects on a PCB, regardless of the complex make up of the target group.

There are many rule types called rule kinds which are: Clearance, ParallelSegment, Width, Length, MatchedLengths, StubLength, PlaneConnect, RoutingTopology, RoutingPriority, RoutingLayers, RoutingCorners, RoutingVias, PlaneClearance, SolderMaskExpansion, PasteMaskExpansion, ShortCircuit, UnRoutedNet, ViasUnderSMD, MaximumViaCount, MinimumAnnularRing, PolygonConnect, AcuteAngle, RoomDefinition, SMDToCorner, ComponentClearance, ComponentOrientations, PermittedLayers, NetsToIgnore, SignalStimulus, OvershootFalling, OvershootRising, UndershootFalling, UndershootRising, MaxMinImpedance, SignalTopValue, SignalBaseValue, FlightTimeRising, FlightTimeFalling, LayerStack, SlopeRising, SlopeFalling, SupplyNets, HoleSize, Testpoint, TestPointUsage, UnConnectedPin, SMDToPlane, SMDNeckDown and LayerPairs pairs.

Common rule attributes for all design rules.

RULEKIND (Rule kind) Specifies the design rule used for a PCB document. The design rules are categorised into 6 groups (Routing Design Rules, Manufacturing Design Rules, High Speed Design Rules, Placement Design Rules, Signal Integrity Design Rules, and Other Rules) for list of attributes for each design rule. See the list below for a description of each design rule (each design rule is identified by the RuleKind parameter in a record of a PCB ASCII file).

NETSCOPE (DifferentNets,SameNetOnly,AnyNet)

LAYERKIND (SameLayer,AdjacentLayers)

SCOPE1Count (Integer) The number of compound design rule scopes for Scope 1.

SCOPE1_x_KIND (String) where X = 0..n For each design rule scope in Scope 1, the kind is specified.

SCOPE1_x_VALUE (String) where X= 0..n For each design rule scope in Scope 1, the value is specified.

SCOPE2COUNT (Integer) The number of compound design rule scopes for Scope 2.

SCOPE2_x_KIND (String) where X = 0..n For each design rule scope in Scope 2, the kind is specified.

SCOPE2_x_VALUE (String) where X = 0..n For each design rule scope in Scope 2, the value is specified.

NAME (String) The defined name for the design rule.

ENABLED (True, False) Denotes whether the defined design rule is enabled or not.

COMMENTLENGTH (Integer) The length of the comment string.

COMMENT (String) The comment string for the design rule. User definable – the comments appear on the Edit Comments page of the Edit Rule dialog.

Routing Design Rules

Clearance constraint rule

Comments: The clearance constraint rule defines the minimum clearance allowed between two primitive objects on a copper layer. This rule is used by the Online and Batch DRC and during autorouting.

GAP (mils) This parameter defines the minimum clearance.

Routing Corners rule

Comments: This corners rule specifies the corner style to be used during autorouting. This rule is used by autorouting.

CORNERSTYLE (90-Degree,45-Degree,Round) The corner style can be a 45 degree chamfer or rounded (using an arc).

MINSETBACK (mils) The minimum setback specifies the minimum distance from the corner location to the start of the corner chamfer or arc.

MAXSETBACK (mils) The maximum setback specifies the minimum distance from the corner location to the start of the corner chamfer or arc.

Routing Layers rule

Comments: This layers rule specifies which layers are to be used during autorouting. This rule is used by autorouting.

LayerX (Not Used,Horizontal,Vertical,Any,1 O'Clock, 2 O'Clock,4 O'Clock,5 O'Clock,45 Up,45 Down). This parameter specifies the direction of routing on these layers.

Where X is one of the following: No Layer, Top Layer, Mid Layer 1, Mid Layer 2, Mid Layer 3,Mid Layer 4, Mid Layer 5, Mid Layer 6,Mid Layer 7, Mid Layer 8, Mid Layer 9, Mid Layer 10, Mid Layer 11, Mid Layer 12, Mid Layer 13, Mid Layer 14, Mid Layer 15, Mid Layer 16, Mid Layer 17, Mid Layer 18, Mid Layer 19, Mid Layer 20, Mid Layer 21, Mid Layer 22, Mid Layer 23, Mid Layer 24, Mid Layer 25, Mid Layer 26, Mid Layer 27, Mid Layer 28, Mid Layer 29, Mid Layer 30, Bottom Layer, Top Overlay, Bottom Overlay, Top Paste, Bottom Paste, Top Solder Mask, Bottom Solder Mask, Internal Plane 1, Internal Plane 2, Internal Plane 3, Internal Plane 4, Internal Plane 5, Internal Plane 6, Internal Plane 7, Internal Plane 8, Internal Plane 9, Internal Plane 10, Internal Plane 11, Internal Plane 12, Internal Plane 13, Internal Plane 14, Internal Plane 15, Internal Plane 16, Drill Guide, Keep Out Layer, Mechanical Layer 1, Mechanical Layer 2, Mechanical Layer 3, Mechanical Layer 4, Mechanical Layer 5, Mechanical Layer 6, Mechanical Layer 7, Mechanical Layer 8,

Mechanical Layer 9,Mechanical Layer 10, Mechanical Layer 11, Mechanical Layer 12, Mechanical Layer 13, Mechanical Layer 14, Mechanical Layer 15,Mechanical Layer 16, Drill Drawing, Multi Layer,BackGround, Connect Layer, DRC Errors, Highlight Layer, Grid Color 1, Grid Color 10, Pad Hole Layer.

Routing Priority rule

Comments: This priority rule specifies routing priorities used to set the order of which nets to be autorouted. This rule is used by autorouting.

PRIORITY (0..100) This parameter assigns a routing priority from 0 to 100. 100 is the highest priority and 0 is the lowest. Routing priorities are relative values which are used to set the order that the nets will be autorouted.

Routing Topology rule

Comments: This topology rule specifies the topology of a net. This rule belongs to the routing category and is used by autorouting.

TOPOLOGY (Shortest,Horizontal,Vertical,Daisy_Simple, Daisy_MidDriven,Daisy_Balanced,Starburst) The topology of a net is the arrangement, or pattern of the pin to pin connections. By default the PCB editor arranges the pin to pin connections of each net to give the shortest overall connection length.

Routing Vias rule

Comments: This routing via style rule specifies the routing via diameter and hole size. This rule belongs to the routing category and is used by autorouting.

HOLEWIDTH (mils) This parameter specifies the hole width of routing vias.

WIDTH (mils) This parameter specifies the width of routing vias.

VIASTYLE (Through Hole, Blind Buried (Adjacent Layers),Blind Buried (Any Layer Pair)) This parameter specifies the style of routing vias.

SMD Neck Down rule

Comments: This SMD Neck Down constraint rule specifies the maximum allowable ratio of the track width to the SMD pad width, expressed as a percentage. Tracks that result in a ratio larger than this amount are tagged as a violation.

PERCENT (real) The percent value specifies the maximum ratio of the track width to the SMD pad width, expressed as a percentage.

SMD To Corner rule

Comments: The SMDToCorner constraint specifies the minimum distance that a track must extend from the narrow edge of a surface mount pad to the first corner. This rule is used by the Online DRC and batch DRC.

DISTANCE (Integer) This parameter specifies the minimum distance.

SMD To Plane rule

Comments: This SMDToPlane constraint specifies the minimum total route length from the center of a surface mount pad to the center of the via or pad connecting to a power plane. This rule is obeyed by the On-line DRC and Batch DRC.

DISTANCE (integer) This parameter specifies the maximum routing length from the SMD pad center to the plane connection pad/via. Typically 0 mils.

Width constraint rule

Comments: This width constraint rule specifies the width of routing tracks and arcs.

MAXLIMIT (mils) This parameter specifies the maximum width of routing tracks and arcs. This value is used for online and batch DRC.

MINLIMIT (mils) This parameter specifies the minimum width of routing tracks and arcs. This value is used for online and batch DRC.

PREFEREDWIDTH (mils) This parameter specifies the preferred width of routing tracks and arcs. This value is used for manual and auto routing.

Manufacturing Design Rules

Acute Angle constraint rule

Comments: The Acute Angle constraint rule controls the problem of overetching of the copper of track corners during manufacturing.

MINIMUM (Real) Denotes the minimum angle for the acute angle rule (at a track corner).

HoleSize constraint rule

Comments: The hole size constraint rule specifies the maximum and minimum hole size, expressed either as exact numeric values, or as a percentage of the pad size.

AbsoluteValues (True,False) If true, the values, MaxLimit and MinLimit are specified. If false, the values MaxPercent and MinPercent are specified.

MaxLimit (mils) This value specifies the maximum hole size for the pad size. Typically 100mils.

MinLimit (mils) This value specifies the minimum hole size for the pad size. Typically 1mils.

MaxPercent (real) This value specifies the maximum percentage of the hole size in respect to the pad size. Typically 80%

MinPercent (real) This value specifies the minimum percentage of the hole size in respect to the pad size. Typically 20%

Layer Pairs rule

Comments: This LayerPairs rule checks to ensure that the used layer-pairs match the current drill-pairs. The used layer-pairs are determined from the vias and pads found in the board, one layer-pair for each Start Layer- End Layer combination that is found.

Enforce(True, False) If true, used layer pairs are matched to the current drill pairs. If false, nothing happens.

Minimum Annular Ring constraint rule

Comments: This MinimumAnnularRing constraint specifies the minimum annular ring allowed on a pad or via. This rule is used by the Online and Batch DRC.

MINIMUMRING (mils) The required minimum annular ring is specified as the difference between the pad radius and the hole radius.

Paste Mask Expansion rule

Comments: The shape that is created on the paste mask layer at each pad site is the pad shape, expanded or contracted radially by the paste-mask expansion rule This rule is used during the output generation.

EXPANSION (mils) The parameter sets the expansion value.

Polygon Connect style rule

Comments: The PolygonConnect Style constraint rule belongs to the manufacturing rules category and is used during a polygon pour.

CONNECTSTYLE (Relief,Direct,NoConnect) The style specifies the style of connection from a component pin to a power plane.

If Relief style is selected, then the following values (Expansion, Entries, Width and air gap) are specified.

RELIEFEXPANSION (mils) Specifies how wide the thermal relief copper connections are.

RELIEFENTRIES (2,4) Specifies the number of thermal relief connections.

POLYGONRELIEFANGLE (45 Angle,90 Angle) Specifies the angle of the connections.

Power Plane Clearance rule

Comments: This power plane clearance rule specifies the clearance between objects belonging to different nets on the power planes. This rule is used during the output generation.

CLEARANCE (mils) The parameter specifies the radial clearance created around vias and pads that pass through but are not connected to a power plane.

Plane Connect style rule

Comments: This Power Plane Connect Style specifies the style of the connection from a component pin to a power plane. This rule is used by the output generation.

PLANECONNECTSTYLE (Relief,Direct,NoConnect) This parameter specifies the style of connection.

If Relief style is selected, then the following values (Expansion, Entries, Width and air gap) are specified.

RELIEFEXPANSION (mils) This parameter specifies the radial width of the expansion measured from the edge of the hole to the edge of the air gap.

RELIEFENTRIES (2,4) This parameter specifies the number of relief entries.

RELIEFCONDUCTORWIDTH (mils) This parameter specifies the gap between reliefs.

RELIEFAIRGAP(mils) This parameter specifies the width of the air gap.

Solder Mask Expansion rule

Comments: The shape that is created on the solder mask layer at each pad and via site is expanded or contracted radially by the solder-mask expansion rule. This rule is used during the output generation.

EXPANSION (mils) When the expansion is a negative value equal to or greater than this specified via radius, then this via is tented. When the expansion value is a negative value equal to or greater than the largest via radius in the PCB design, all vias are tented.

Test point style rule

Comments: The testpoint style rule specifies the allowable physical parameters of pads and vias that are flagged as testpoints. The Find Testpoint feature and the Autorouter use the Allowed Side settings in the following order of preference (highest to lowest): Bottom (surface mount), Top (surface mount), Bottom via, Top via, Bottom thru-hole, Top thru-hole. This rule is obeyed by the Find Testpoint feature, the autorouter, and the on-line and batch DRC.

SIDE (Integer). The SIDE parameter is a bitmask where there are four side values that can be ("OR"ed) combined individually (Top = 1, Bottom =2, Thru-Hole Top =4, Thru-Hole Bottom=8). For example, the Top and Bottom allowed sides have a value of 3, and this is assigned to the SIDE parameter.

MINSIZE (mils) This value specifies the minimum size Typically 40 mils.

MAXSIZE (mils) This value specifies the maximum size Typically 100 mils.

PREFEREDSIZE (mils) Typically 60 mils. This is used for automatic generation of testpoints.

MINHOLESIZE (mils) Typically 20 mils.

MAXHOLESIZE (mils) Typically 40 mils.

PREFEREDHOLESIZE (mils) Typically 32 mils. This is used for automatic generation of testpoints.

TESTPOINTGRID (mils) Typically 1 mils.

Test Point Usage rule

Comments: The Testpoint usage rule specifies which nets require a testpoint. The DRC report is used to identify each net that fails this rule, and the Testpoint report feature in the CAM Manager is used to identify the location of valid testpoints. This rule is obeyed by the Find Testpoint feature, the autorouter, and the on-line and batch DRC.

VALID (0,1,2) 0 = Require, 1 = Invalid, 2 =Ignore

ALLOWMULTIPLE (True,False)

High Speed Design Rules

Daisy Chain Stub Length rule

Comments: This Daisy Chain Stub Length rule specifies the maximum permissible stub length for a net with a daisy chain topology.

LIMIT (mils) This parameter specifies the maximum permissible stub length.

Length constraint rule

Comments: The Length Constraint rule specifies the minimum and maximum lengths of a net. This rule is used by the Online and Batch DRC.

MAXLIMIT (mils) This parameter specifies the maximum length of a net.

MINLIMIT (mils) This parameter specifies the minimum length of a net.

Matched Lengths rule

Comments: The matched lengths rule specifies the degree to which nets can have different lengths. The longest net is located (based on the rule scope) and its length is compared to each of the other nets specified by the scope. The PCB editor can add "accordion" sections to nets to equalize their lengths. The matched Net Lengths rule is used by the Online and Batch DRC.

TOLERANCE (mils) This parameter specifies the length of nets which fail the matched length requirements.

STYLE (90-Degree, 45-Degree, Round) The accordion style. The 90 degree style is the most compact and the rounded style is the least compact.

AMPLITUDE (mils) This parameter specifies the amplitude of an accordion section.

GAP (mils) If Style = 90 Degrees or 45 degrees, the gap of an accordion section is specified.

Maximum Via Count constraint rule

Comments: The maximum via count constraint specifies the maximum number of vias permitted on each net covered by the rule scope. This rule is used by the Online and Batch DRC.

LIMIT (integer) Specifies the maximum number of vias permitted.

Parallel Segment rule

Comments: The parallel segment rule specifies the distance two track segments can run in parallel for a given separation. This rule is used by the Online and Batch DRC.

LIMIT (mils) This parameter specifies the distance two track segments can run in parallel.

GAP (mils) This parameter specifies the gap between two parallel track segments.

Vias Under SMD constraint rule

Comments: The ViasUnderSMD constraint rule specifies whether vias can be placed under SMD pads during autorouting. This rule is used by the Online and Batch DRC.

ALLOWED (True, False) Specifies whether vias can be placed under SMD pads during auto routing.

Placement Design Rules

Component Clearance constraint rule

Comments: The ComponentClearance constraint belongs to the Placement Rule category. This rule is used during auto placing with the Cluster Placer.

GAP (mils) Specifies the minimum distance that components must be from each other.

COLLISIONCHECKMODE (0,1,2) 0 = Quick Check, 1 = MultiLayerCheck, 2 = FullCheck.

Component Orientations rule

Comments: The Component Orientations rule specifies allowable component orientations, allowing the Cluster autoplacer to use any of the enabled orientations. This rule belongs to the Placement Rule category and is used during auto placing with the Cluster Placer.

ALLOWEDROTATIONS (Integer) A bitmask value that which is a combination ('ORed') of the following values ($01, $02, $08, $10, $20). Specifies allowable component orientations (0 Degrees, 90 degrees, 180 degrees, 270 degrees) and multiple orientations are permitted.

Nets To Ignore rule

Comments: The NetsToIgnore rule defines which nets should be ignored during autoplacing with the cluster placer. Ignoring power nets can assist in placement speed and quality. This rule is belongs to the Placement category and is used during auto placing with the Cluster Placer.

Permitted Layers rule

Comments: This permitted layers rule specifies the layers that the components can be placed on. When the cluster placer is started each component has its layer set to satisfy the permitted layer rules. This rule is used during auto placing with the Cluster Placer.

TOPLAYERPERMITTED (True, False) Specifies whether the components can be placed on the top layer or not.

BOTTOMLAYERPERMITTED (True, False) Specifies whether the components can be placed on the bottom layer or not.

Room Definition rule

Comments: A room definition defines a rectangular region which the specified objects must be either kept in or kept out of.

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (Layer) The layer where a room object exists on.

LOCKED (True, False) Denotes whether a room object can be locked or not so the autorouter server cannot move this object.

REGION_X1 (mils) The initial X of the corner of a room object.

REGION_Y1 (mils) The initial Y of the corner of a room object.

REGION_X2 (mils) The final X of the corner of a room object.

REGION_Y2 (mils) The final Y of the corner of a room object.

CONFINEMENTSTYLE (ConfineIn,ConfineOut) Specifies a region where specified objects are either allowed in or not allowed in.

Signal Integrity Design Rules

Flight Time Falling

Comments: Flight time is the signal delay time introduced by the interconnect structure. It is calculated as the time it takes to drive the actual input to the threshold voltage, less the time it would take to drive a reference load (connected directly to the output) to the threshold voltage. This rule is used by signal integrity analysis.

MAXIMUM (extended) This value specifies the maximum allowable flight time on signal falling edge. Typically 1.000n seconds.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the notes in the Units section.

Flight Time Rising

Comments: Flight time is the signal delay time introduced by the interconnect structure. It is calculated as the time it takes to drive the actual input to the threshold voltage, less the time it would take to drive a reference load (connected directly to the output) to the threshold voltage. This rule is used by signal integrity analysis.

MAXIMUM (extended) This value specifies the maximum allowable flight time on signal rising edge. Typically 1.000n seconds.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Max Min Impedance

Comments: Net impedance is a function of the conductor geometry and conductivity, the surrounding dielectric material (the board base material, multi-layer insulation, solder mask, etc) and the physical geometry of the board (distance to other conductors in the z-plane). This rule is used by signal integrity analysis.

MINIMUM (extended) Specifies the minimum net impedance allowed.

MAXIMUM (extended) Specifies the maximum net impedance allowed.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Overshoot Falling

Comments: This rule is used by signal integrity analysis.

MAXIMUM (extended) Specify the maximum allowable overshoot on the falling edge of the signal.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Overshoot Rising

Comments: This rule is used by signal integrity analysis.

MAXIMUM (extended) Specify the maximum allowable overshoot on the rising edge of the signal.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Signal Base Value

Comments: The base value is the voltage that a signal settles to in the low state. This rule is used by signal integrity analysis.

MAXIMUM (extended) This value specifies the maximum allowable base value. Typically 0.000 volts.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Signal Stimulus

Comments: The Signal Stimulus rule specifies the characteristics of the stimulus signal that is used on the signal integrity analysis.

KIND (0,1,2) 0 = Constant Level, 1= Single pulse , 2= Periodic pulse

LEVEL (0,1) 0 = Low Level, 1= High Level

STARTTIME (extended). The starting time of the stimulus signal.

STOPTIME (extended The final time of the stimulus signal.

PERIODTIME (extended) The period time of a stimulus signal.

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Signal Top Value

Comments: The top value is the voltage that a signal settles to in the high state. This rule is used by signal integrity analysis.

MINIMUM (extended) This value specifies the minimum allowable top value. Typically 5.000 volts.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

SlopeFalling

Comments: Falling edge slope is the time it takes for a signal to fall from the threshold voltage (VT), to a valid low (VIL). This rule is used by signal integrity analysis.

MAXIMUM (extended) This value specifies the maximum allowable slope time. Typically 1.000n seconds maximum.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Slope Rising

Comments: The rising edge slope is the time it takes for a signal to rise from the threshold voltage (VT), to a valid high (VIH). This rule is used by signal integrity analysis.

MAXIMUM (extended) This value specifies the maximum allowable slope time. Typically 1.000n seconds maximum.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Supply Nets

Comments: The signal integrity tool needs to know each supply net name and voltage on a PCB. This rule is used by signal integrity analysis.

VOLTAGE (extended) This value Identifies a supply net and specifies its voltage (or set of nets using the net class scope). Typically 0.000 volts.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Undershoot Falling

Comments: This rule is used by signal integrity analysis.

MAXIMUM (extended) Specify the maximum allowable undershoot on the falling edge of the signal.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Undershoot Rising

Comments: This rule is used by signal integrity analysis.

MAXIMUM (extended) Specify the maximum allowable undershoot on the rising edge of the signal.

Notes:

The extended value can have a suffix (T, G, MEG, K, U, N, P, F). See the Units used in the document section.

Other Design Rules

Short Circuit rule

Comments: A short circuit exists if two objects that have different net names touch.

ALLOWED (True, False) If true, short circuits of nets are allowed. It is used for example, tying two ground nets together. If false, short circuit nets are not allowed.

Unconnected Pin constraint rule

Comments: The unconnected pin constraint rule detects pins that have no net assigned and no connecting tracks. This rule is obeyed by the On-line and batch DRC.

UnRouted Net constraint rule

Comments. The un-routed net constraint tests the completion status of each net identified by the scope. A un-routed net constraint rule belongs to the Others category.

Typical Rule object record

```
RECORD=Rule|RULEKIND=ShortCircuit|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1_
0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Shor
tCircuit|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |ALLOWED=FALSE

RECORD=Rule|RULEKIND=UnRoutedNet|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1_0
_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=UnRou
tedNet|ENABLED=TRUE|COMMENTLENGTH=0|Comment=

RECORD=Rule|RULEKIND=Clearance|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1_0_K
IND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Clearan
ce|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |GAP= 1.00000000000000E+0001mil

RECORD=Rule|RULEKIND=PlaneConnect|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1_
0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Plan
eConnect|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |PLANECONNECTSTYLE=Relief|RELIEFEXPANSION=
2.00000000000000E+0001mil|RELIEFENTRIES=4|RELIEFCONDUCTORWIDTH=
1.00000000000000E+0001mil|RELIEFAIRGAP= 1.00000000000000E+0001mil

RECORD=Rule|RULEKIND=PlaneClearance|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE
1_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Pl
aneClearance|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |CLEARANCE= 2.00000000000000E+0001mil

RECORD=Rule|RULEKIND=PolygonConnect|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE
1_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Po
lygonConnect|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |CONNECTSTYLE=Relief|RELIEFCONDUCTORWIDTH=
1.00000000000000E+0001mil|RELIEFENTRIES=4|POLYGONRELIEFANGLE=90 Angle

RECORD=Rule|RULEKIND=Width|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1_0_KIND=
Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Width|ENABL
ED=TRUE|COMMENTLENGTH=0|Comment= |MAXLIMIT= 1.00000000000000E+0001mil|MINLIMIT=
1.00000000000000E+0001mil|PREFEREDWIDTH= 1.00000000000000E+0001mil

RECORD=Rule|RULEKIND=RoutingPriority|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOP
E1_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=R
outingPriority|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |PRIORITY=0

RECORD=Rule|RULEKIND=RoutingLayers|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1
_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Rou
tingLayers|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |TOP LAYER=Horizontal|MID LAYER 1=Not Used|MID
LAYER 2=Not Used|MID LAYER 3=Not Used|MID LAYER 4=Not Used|MID LAYER 5=Not Used|MID LAYER 6=Not
Used|MID LAYER 7=Not Used|MID LAYER 8=Not Used|MID LAYER 9=Not Used|MID LAYER 10=Not Used|MID
LAYER 11=Not Used|MID LAYER 12=Not Used|MID LAYER 13=Not Used|MID LAYER 14=Not Used|MID LAYER
15=Not Used|MID LAYER 16=Not Used|MID LAYER 17=Not Used|MID LAYER 18=Not Used|MID LAYER 19=Not
Used|MID LAYER 20=Not Used|MID LAYER 21=Not Used|MID LAYER 22=Not Used|MID LAYER 23=Not Used|MID
LAYER 24=Not Used|MID LAYER 25=Not Used|MID LAYER 26=Not Used|MID LAYER 27=Not Used|MID LAYER
28=Not Used|MID LAYER 29=Not Used|MID LAYER 30=Not Used|BOTTOM LAYER=Vertical

RECORD=Rule|RULEKIND=RoutingCorners|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE
1_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Ro
utingCorners|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |CORNERSTYLE=45-Degree|MINSETBACK=
1.00000000000000E+0002mil|MAXSETBACK= 1.00000000000000E+0002mil

RECORD=Rule|RULEKIND=RoutingVias|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1_0
_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Routi
ngVias|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |HOLEWIDTH= 2.80000000000000E+0001mil|WIDTH=
5.00000000000000E+0001mil|VIASTYLE=Through Hole|MINHOLEWIDTH= 2.80000000000000E+0001mil|MINWIDTH=
5.00000000000000E+0001mil|MAXHOLEWIDTH= 2.80000000000000E+0001mil|MAXWIDTH=
5.00000000000000E+0001mil

RECORD=Rule|RULEKIND=PasteMaskExpansion|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|S
COPE1_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAM
E=PasteMaskExpansion|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |EXPANSION= 0.00000000000000E+0000mil

RECORD=Rule|RULEKIND=SolderMaskExpansion|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|
SCOPE1_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NA
ME=SolderMaskExpansion|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |EXPANSION= 4.00000000000000E+0000mil

RECORD=Rule|RULEKIND=RoutingTopology|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOP
E1_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=R
outingTopology|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |TOPOLOGY=Shortest

RECORD=Rule|RULEKIND=TestPointUsage|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE
1_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Te
stPointUsage|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |VALID=0|ALLOWMULTIPLE=FALSE

RECORD=Rule|RULEKIND=Testpoint|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1_0_K
IND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=Testpoi
nt|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |SIDE=15|MINSIZE= 4.00000000000000E+0001mil|MAXSIZE=
1.00000000000000E+0002mil|PREFEREDSIZE= 6.00000000000000E+0001mil|MINHOLESIZE=
2.00000000000000E+0001mil|MAXHOLESIZE= 4.00000000000000E+0001mil|PREFEREDHOLESIZE=
3.20000000000000E+0001mil|TESTPOINTGRID= 1.00000000000000E+0000mil
```

```
RECORD=Rule|RULEKIND=ComponentClearance|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|S
COPE1_0_KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAM
E=ComponentClearance|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |GAP=
1.00000000000000E+0001mil|COLLISIONCHECKMODE=0

RECORD=Rule|RULEKIND=LayerPairs|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1_0_
KIND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=LayerP
airs|ENABLED=TRUE|COMMENTLENGTH=0|Comment= |ENFORCE=TRUE

RECORD=Rule|RULEKIND=HoleSize|NETSCOPE=DifferentNets|LAYERKIND=SameLayer|SCOPE1COUNT=1|SCOPE1_0_KI
ND=Board|SCOPE1_0_VALUE=Board|SCOPE2COUNT=1|SCOPE2_0_KIND=Board|SCOPE2_0_VALUE=Board|NAME=HoleSize
 |ENABLED=TRUE|COMMENTLENGTH=0|Comment= |ABSOLUTEVALUES=TRUE|MAXLIMIT=
1.00000000000000E+0002mil|MINLIMIT= 1.00000000000000E+0000mil|MAXPERCENT=80.000|MINPERCENT=20.000
```

Connection Objects

The pin-to-pin connections in each net are displayed as a series of thin connection lines. The line that connects each pin in the net to another pin in the net is called a From-To, going FROM one pin in the net TO another pin. The From-Tos are collectively referred to as the Ratsnest. Please note that in Protel API documentation, a connection is actually referred as a FromTo.

Connection attributes

NET (Integer) A connection belongs to a net and each net is identified by its number.

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LOCKED (True, False) Denotes whether a connection object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if a connection is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether a connection has been routed manually.

X1 (Mils) Denotes the starting X location of a connection.

Y1 (Mils) Denotes the starting Y location of a connection.

X2 (Mils) Denotes the final X location of a connection.

Y2 (Mils) Denotes the final Y location of a connection.

LAYER1 (String) The first layer which the starting point of the connection is on.

LAYER2 (String) The second layer which the end point of the connection is on.

Typical Connection object record

```
|RECORD=Connection|NET=45|SELECTION=FALSE|LAYER=MULTILAYER|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERR
OUTED=TRUE|X1= 8.24200000000000E+0003mil|Y1= 7.14200000000000E+0003mil|X2=
8.63500000000000E+0003mil|Y2= 6.73500000000000E+0003mil|LAYER1=TOP|LAYER2=TOP
```

Embedded Objects

The PCB embedded objects are embedded within a PCB document. When you save a PCB document, the embedded objects are saved as well. There is no limit on how many embedded objects you can store within a PCB document, it is only subject to the amount of memory your computer has. Embedded objects are invisible, thus you cannot use normal PCB tools to manipulate embedded objects. You as a programmer can use embedded objects for your specific programming needs.

Embedded attributes

Name (string) Denotes the name string.

User defined parameters and values can be set via the PCB API. In this case, see the record below, LayerSetname, LayerCount, Layer1, Layer 2 are user defined parameters.

Typical embedded object record

```
|RECORD=Embedded|NAME=Layer Viewer|LayerSetName=Custom set|LayerCount=2|Layer1=Bottom Solder
Mask|Layer2=Top Paste
```

Arc Objects

Arcs are essentially circular track segments. They can be placed on any layer with any radius and width from 0.001 to 10000 mils wide. The angular resolution is 0.001 degree. Arcs are also used when the PCB editor generates polygon fills.

Arc attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which an arc is on.

LOCKED (True, False) Denotes whether an arc object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if an arc is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether an arc has been routed manually.

LOCATION.X (mils) The X location of an arc object.

LOCATION.Y (mils) The Y location of an arc object.

RADIUS (mils) Denotes the radius of an arc object in mils.

STARTANGLE (TReal value 0 – 360 degrees.) Denotes the initial angle of an arc object in degrees.

ENDANGLE (TReal value 0 – 360 degrees.) Denotes the final angle of an arc object in degrees.

WIDTH (mils) Denotes the line width of the arc in mils.

SUBPOLYINDEX (Integer) If the SubPolyIndex = 0, this arc object is a standalone arc. If not zero, it is part of a polygon object.

Typical arc object record

```
|RECORD=Arc|SELECTION=FALSE|LAYER=TOP|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE|LOCATION.X
= 1.22200000000000E+0004mil|LOCATION.Y= 1.26200000000000E+0004mil|RADIUS=
1.49264870000000E+0003mil|STARTANGLE=0.000|ENDANGLE=360.000|WIDTH=
1.00000000000000E+0001mil|SUBPOLYINDEX=0
```

Pad objects

Pads can be either multi-layered or placed on any individual layer. For surface mount components and edge connectors, single layer pads are placed on the Top and/or Bottom layers. Pads shapes include circular, rectangular, rounded rectangular or octagonal with X and Y size definable by the user from 1 to 10000 mils. Hole size can range from 0 (SMD) to 1000 mils. Pads can be identified with a designator. On multi-layer pads, the top layer, mid-layers and bottom layer pad shape and size can be independently assigned to define pad stacks. Pads can be used individually as free pads or they can be incorporated with other primitives into components.

Pad attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a pad is on.

LOCKED (True, False) Denotes whether a pad object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if a pad is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether a pad has been routed manually.

NAME (String). The pattern name of the component.

X (mils) The location of a pad at X coordinate.

Y (mils) The location of a pad at Y coordinate.

XSIZE (mils) The horizontal size of a pad.

YSIZE (mils) The vertical size of a pad.

SHAPE (ROUND,RECTANGLE,OCTAGONAL). Denotes the shape of the pad.

HOLESIZE (mils) The size of the hole of a pad.

ROTATION (TReal value 0 – 360 degrees.) Denotes the rotation of a pad in degrees.

PLATED (True,False) This determines whether pad hole is plated or not.

DAISYCHAIN (Source,Terminator,Load) Denotes the daisy chain type for the pad object. High speed designs require that signal reflections must be minimized. To achieve this the high speed nets are arranged with a daisy chain topology, where all the pins are connected one after the other, with the source pin at one end and a terminator pin at the other end of the chain.

CSSV (0,1) Denotes whether the Plane Connection Style is valid or not. Derived from the pad cache in the PCB editor.

CPLV (0,1) Denotes whether the Planes is valid or not. Derived from the pad cache in the PCB editor.

CCWV (0,1) Denotes whether the Relief Conductor Width is valid or not. Derived from the pad cache in the PCB editor.

CENV (0,1) Denotes whether the Relief Entries is valid or not. Derived from the pad cache in the PCB editor.

CAGV (0,1) Denotes whether the Relief Air Gap is valid or not. Derived from the pad cache in the PCB editor.

CPEV (0,1) Denotes whether the Paste Mask Expansion is valid or not. Derived from the pad cache in the PCB editor.

CSEV (0,1) Denotes whether the Solder Mask Expansion is valid or not. Derived from the pad cache in the PCB editor.

CPCV (0,1) Denotes whether the Power Plane Clearance is valid or not. Derived from the pad cache in the PCB editor.

CPRV (0,1) Denotes whether the Power Plane Relief Expansion is valid or not. Derived from the pad cache in the PCB editor.

CCS (NoConnect, Relief, Direct) Denotes the plane connection style. There are styles of connection from a component pin to a polygon plane. Two connection methods are available, direct connections (solid copper to the pin) or thermal relief connections. If Relief Connect is selected then how wide the thermal relief copper connections are; the number of connections; and the angle of the connections.

CPL (Integer) Denotes the number of planes.

CCW (mils) Denotes the relief conductor width.

CEN (Integer) Denotes the number of relief entries.

CAG (mils) Denotes the relief air gap in mils.

CPE (mils) Denotes the paste mask expansion in mils.

CSE (mils) Denotes the solder mask expansion in mils.

CPC (mils) Denotes the power plane clearance in mils.

CPR (mils) Denotes the power plane relief expansion in mils.

Typical Pad Object record

```
|RECORD=Pad|SELECTION=FALSE|LAYER=MULTILAYER|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE|NAM
E=4|X= 3.52000000000000E+0003mil|Y= 8.22000000000000E+0003mil|XSIZE=
6.00000000000000E+0001mil|YSIZE= 6.00000000000000E+0001mil|SHAPE=ROUND|HOLESIZE=
3.00000000000000E+0001mil|ROTATION=0.000|PLATED=TRUE|DAISYCHAIN=Load|CCSV=0|CPLV=0|CCWV=1|CENV=1|C
AGV=1|CPEV=1|CSEV=1|CPCV=1|CPRV=1|CCW= 1.00000000000000E+0001mil|CEN=4|CAG=
1.00000000000000E+0001mil|CPE= 0.00000000000000E+0000mil|CSE= 4.00000000000000E+0000mil|CPC=
2.00000000000000E+0001mil|CPR= 2.00000000000000E+0001mil
```

Via Objects

Vias are either multi-layer, blind or buried and can be any diameter from 2 to 10000 mils wide. The hole size can be set from 0 to 1000 mils. A multi-layer via passes from the Top layer to the bottom layer and allows connections to all internal signal layers. A blind via connects from the surface of the board to an internal layer, a buried via connects from one internal layer to another internal layer.

Via attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a via is on.

LOCKED (True, False) Denotes whether the via object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if the via is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether the track has been routed manually.

X (Mils) The X location of a Via object in mils.

Y (Mils) The Y location of a Via object in mils.

DIAMETER (Mils) The size of a via object in mils.

HOLESIZE (Mils) The hole size of a via object in mils.

STARTLAYER (string) Denotes the starting layer of the via object. STARTLAYER is the higher layer of the stack = Top.

ENDLAYER (string) Denotes the ending layer of the via object. ENDLAYER is lower layer of the stack = Bottom.

CCSV (0,1) Denotes whether the Plane Connection Style is valid or not. Derived from the pad cache in the PCB editor.

CPLV (0,1) Denotes whether the Planes is valid or not. Derived from the pad cache in the PCB editor.

CCWV (0,1) Denotes whether the Relief Conductor Width is valid or not. Derived from the pad cache in the PCB editor.

CENV (0,1) Denotes whether the Relief Entries is valid or not. Derived from the pad cache in the PCB editor.

CAGV (0,1) Denotes whether the Relief Air Gap is valid or not. Derived from the pad cache in the PCB editor.

CPEV (0,1) Denotes whether the Paste Mask Expansion is valid or not. Derived from the pad cache in the PCB editor.

CSEV (0,1) Denotes whether the Solder Mask Expansion is valid or not. Derived from the pad cache in the PCB editor.

CPCV (0,1) Denotes whether the Power Plane Clearance is valid or not. Derived from the pad cache in the PCB editor.

CPRV (0,1) Denotes whether the Power Plane Relief Expansion is valid or not. Derived from the pad cache in the PCB editor.

CCS (NoConnect, Relief, Direct) Denotes the plane connection style.

CPL (Integer) Denotes the number of planes.

CCW (mils) Denotes the relief conductor width.

CEN (Integer) Denotes the number of relief entries.

CAG (mils) Denotes the relief air gap in mils.

CPE (mils) Denotes the paste mask expansion in mils.

CSE (mils) Denotes the solder mask expansion in mils.

CPC (mils) Denotes the power plane clearance in mils.

CPR (mils) Denotes the power plane relief expansion in mils.

Notes:

Vias and pads share the same pad cache, because they share some of the same attributes.

Typical Via Object Record

```
|RECORD=Via|SELECTION=FALSE|LAYER=MULTILAYER|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE|X=
3.56000000000000E+0003mil|Y= 8.24000000000000E+0003mil|DIAMETER=
5.00000000000000E+0001mil|HOLESIZE=
2.80000000000000E+0001mil|STARTLAYER=TOP|ENDLAYER=BOTTOM|CCSV=0|CPLV=0|CCWV=1|CENV=1|CAGV=1|CPEV=0
|CSEV=1|CPCV=1|CPRV=1|CCW= 1.00000000000000E+0001mil|CEN=4|CAG= 1.00000000000000E+0001mil|CSE=
4.00000000000000E+0000mil|CPC= 2.00000000000000E+0001mil|CPR= 2.00000000000000E+0001mil
```

Track Objects

Tracks are placed on any layer using any width from 0.001 to 10000 mils wide. Tracks are used to route connections on signal layers, to define the board outline on a mechanical layer, to create a component outline on the silkscreen layer, or to define keep out regions on the keepout layer. Tracks are also used to generate polygon fills.

Track attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a track is on.

LOCKED (True, False) Denotes whether the track object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if the track is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether the track has been routed manually.

X1 (Mils) Denotes the starting X location of a track.

Y1 (Mils) Denotes the starting Y location of a track.

X2 (Mils) Denotes the final X location of a track.

Y2 (Mils) Denotes the final Y location of a track.

WIDTH (Mils) Denotes the width of a track.

SUBPOLYINDEX (integer) If the SubPolyIndex = 0, the track object is a standalone track. If not zero, it is part of a polygon object.

Typical Track Object Record

```
|RECORD=Track|POLYGON=0|SELECTION=FALSE|LAYER=BOTTOMSOLDER|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERR
OUTED=TRUE|X1= 6.18000000000000E+0003mil|Y1= 1.19000000000000E+0004mil|X2=
6.32000000000000E+0003mil|Y2= 1.19000000000000E+0004mil|WIDTH=
8.00000000000000E+0000mil|SUBPOLYINDEX=0
```

Text Objects

Text strings can be placed on any layer of a PCB document with any height from 0.01 to 10000 mils.

Text attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a text object is on.

LOCKED (True, False) Denotes whether a text object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if a text object is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether a text has been routed manually.

X (Mils) Denotes the X location of a text.

Y (Mils) Denotes the Y location of a text.

HEIGHT (mils) Denotes the height of the text object. Default is 60 mils and the range is 0.01 mils to 10000 mils.

FONT (DEFAULT,SANS SERIF,SERIF) Determines the font style of the text object.

ROTATION (TReal value 0 – 360 degrees.) Denotes the rotation of the text object in degrees. Minimum angular resolution is 0.001 degrees.

MIRROR (True,False) Denotes whether the text is mirrored or not. By default a text object is not mirrored.

TEXT (string) Denotes the string of the text object.

WIDTH (mils) Denotes the current text stroke width of the string which defines the thickness of the lines used to produce the lettering Default is 10 mils and the range is 0.001 mils to 255 mils.

Typical Text Object Record

```
|RECORD=Text|SELECTION=FALSE|LAYER=BOTTOM|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE|X=
3.38000000000000E+0003mil|Y= 8.74000000000000E+0003mil|HEIGHT=
6.00000000000000E+0001mil|FONT=DEFAULT|ROTATION=0.000|MIRROR=FALSE|TEXT=Testing|WIDTH=
1.00000000000000E+0001mil
```

Fill Objects

Rectangular fills are placed to indicate solid copper areas of any size on any signal layer. When placed on internal plane, solder or paste mask layers, fills are used to designate voids.

Fill attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a fill is on.

LOCKED (True, False) Denotes whether a fill object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if a fill is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether a fill has been routed manually or not.

X1 (Mils) Denotes the starting X location of a fill.

Y1 (Mils) Denotes the starting Ylocation of a fill.

X2 (Mils) Denotes the final X location of a fill.

Y2 (Mils) Denotes the final Y location of a fill.

ROTATION (TReal value 0 – 360 degrees.)

Typical Fill Object Record

```
|RECORD=Fill|SELECTION=FALSE|LAYER=TOP|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE|X1=
1.07600000000000E+0004mil|Y1= 5.00000000000000E+0003mil|X2= 1.53400000000000E+0004mil|Y2=
7.12000000000000E+0003mil|ROTATION=0.000
```

From-To Objects

A from-to object is a user defined connection in a net going from a specific pad to another specific pad.

From-To attributes

SELECTION (True, False). Selected objects can be cut or copied to the clipboard. They can then be pasted elsewhere onto the current document or into another PCB document.

LAYER (String) The layer which a From-To is on.

LOCKED (True, False) Denotes whether a fill object can be locked so the autorouter server cannot move this object.

POLYGONOUTLINE (True,False) Denotes if a From-To is used for the outline of a polygon.

USERROUTED (True,False) Denotes whether a From-To has been routed manually or not.

FROM (string) The initial node (pad) of a connection.

TO (string) The final node (pad) of a connection.

NET (string) The net where this specified FromTo is part of.

Typical From-To Object Record

```
RECORD=FromTo|SELECTION=FALSE|LAYER=TOP|LOCKED=FALSE|POLYGONOUTLINE=FALSE|USERROUTED=TRUE|From=U12
-2|To=SW1-6|Net=4800
```