LEF/DEF Language Reference

Product Version 5.6
June 2004

Contents

Preface

This manual is a language reference for users of the Cadence® Library Exchange Format (LEF) and Design Exchange Format (DEF) integrated circuit (IC) description languages.

LEF defines the elements of an IC process technology and associated library of cell models. DEF defines the elements of an IC design relevant to physical layout, including the netlist and design constraints. LEF and DEF inputs are in ASCII form.

This manual assumes that you are familiar with the development and design of integrated circuits.

This preface provides the following information:

- What's New on page 7
- Typographic and Syntax Conventions on page 7
- Character Information on page 8

What's New

For information on what is new or changed in LEF and DEF for version 5.6 see *What's New in LEF/DEF*.

Typographic and Syntax Conventions

This list describes the conventions used in this manual.

`text`	Words in `monospace` type indicate keywords that you must enter literally. These keywords represent language tokens.
`variable`	Words in *`italics`* indicate user-defined information for which you must substitute a name or a value.
`objRegExpr`	An object name with the identifier *`objRegExpr`* represents a regular expression for the object name.

`pt`	Represents a point in the design. This value corresponds to a coordinate pair, such as x y. You must enclose a point within parentheses, with space between the parentheses and the coordinates. For example, `RECT ( 1000 2000 ) ( 1500 400 ).`
`|`	Vertical bars separate possible choices for a single argument. They take precedence over any other character.
`[ ]`	Brackets denote optional arguments. When used with vertical bars, they enclose a list of choices from which you can choose one.
`{ } ...`	Braces followed by three dots indicate that you must specify the argument at least once, but you can specify it multiple times.
`{ }`	Braces used with vertical bars enclose a list of choices from which you must choose one.
`...`	Three dots indicate that you can repeat the previous argument. If they are used with brackets, you can specify zero or more arguments. If they are used with braces, you must specify at least one argument, but you can specify more.
`,...`	A comma and three dots together indicate that if you specify more than one argument, you must separate those arguments with commas.
`" "`	Quotation marks enclose string values. Write quotation marks within a string as `\"`. Write a backslash within a string as `\\`.

Any characters not included in the list above are required by the language and must be entered literally.

Character Information

LEF and DEF support the following characters:

!	< and >
$	. (period)
&	?

`|`	`{ }`
`:`	`’` (single quotation mark)
`‘`	`“` (double quotation mark)
`@`	_ (underbar)
`~`	^
`=`	`,` (comma)
Uppercase and lowercase alphabet characters	Numbers

DEF reserves the following characters for special functions:

`( )`	Coordinates
`+`	Start of new keyword
`-`	Coordinates and start of new keyword
`[ ]`	Default special characters for bus bits unless overridden by `BUSBITCHARS`
`/`	Default special character for hierarchy unless overridden by `DIVIDERCHAR`

LEF and DEF names cannot contain the following ASCII characters:

`\n`	Newline
	Space
`;`	Semicolon

Note: LEF and DEF names also cannot contain the ASCII character used by the place-and-route tool for comments. For example, if the tool uses the pound sign (#) for comments, it cannot be used in a LEF or DEF name.

LEF and DEF interpret the following characters as regular expressions. Use these characters in LEF and DEF names only when you intend a regular expression.

`*`	Asterisk	Matches any sequence of characters
`%`	Percent	Matches any single character

Note: Pattern matching only works in a few areas of the DEF file, such as component name matching in the `SPECIALNET` section, and component name matching in `GROUP` definitions.

You can use the backslash (\) as an escape character. When the backslash precedes a character that has special meaning in LEF or DEF, the special meaning of the character is ignored. The following characters have special meanings that can be escaped:

`BUSBITCHARS` *`delimiterPair`*	The characters you specify to enclose bus bits
`DIVIDERCHAR` *`character`*	The character you specify to express hierarchy
`\`	The backslash character

Note: You cannot use the escape character with the pound sign (#).

1

LEF Syntax

This chapter contains information about the following topics:

About Library Exchange Format Files

A Library Exchange Format (LEF) file contains library information for a class of designs. Library data includes layer, via, placement site type, and macro cell definitions. The LEF file is an ASCII representation using the syntax conventions described in "Typographic and Syntax Conventions" on page 7.

General Rules

Note the following information about creating LEF files:

- Lines in the LEF file are limited to 2,048 characters (extra characters are truncated).
- Distance is specified in microns.
- Distance precision is controlled by the `UNITS` statement.
- LEF statements end with a semicolon (`;`). You must leave a space between the last character in the statement and the semicolon.

Managing LEF Files

You can define all of your library information in a single LEF file; however this creates a large file that can be complex and hard to manage. Instead, you can divide the information into two files, a "technology" LEF file and a "cell library" LEF file.

A technology LEF file contains all of the LEF technology information for a design, such as placement and routing design rules, and process information for layers. A technology LEF file can include any of the following LEF statements:

```
[VERSION statement]
[BUSBITCHARS statement]
[DIVIDERCHAR statement]
[UNITS statement]
[MANUFACTURINGGRID statement]
[USEMINSPACING statement]
[CLEARANCEMEASURE statement ;]
[PROPERTYDEFINITIONS statement]
[LAYER (Nonrouting) statement
 | LAYER (Routing) statement] ...
[SPACING statement ]
[MAXVIASTACK statement]
[VIA statement] ...
[VIARULE statement] ...
[VIARULE GENERATE statement] ...
[NONDEFAULTRULE statement] ...
[SITE statement] ...
[BEGINEXT statement] ...
[END LIBRARY]
```

A cell library LEF file contains the macro and standard cell information for a design. A library LEF file can include any of the following statements:

```
[VERSION statement]
[BUSBITCHARS statement]
[DIVIDERCHAR statement]
[VIA statement] ...
[SITE statement]
[MACRO statement
  [PIN statement] ...
  [OBS statement ...] ] ...
[BEGINEXT statement] ...
[END LIBRARY]
```

When reading in LEF files, always read in the technology LEF file first.

Order of LEF Statements

LEF files can contain the following statements. You can specify statements in any order; however, data must be defined before it is used. For example, the `UNITS` statement must be defined before any statements that use values that are dependent on `UNITS` values, `LAYER` statements must be defined before statements that use the layer names, and `VIA` statements must be defined before referencing them in other statements. If you specify statements in the following order, all data is defined before being used.

```
[VERSION statement]
[BUSBITCHARS statement]
[DIVIDERCHAR statement]
[UNITS statement]
[MANUFACTURINGGRID statement]
[USEMINSPACING statement]
[CLEARANCEMEASURE statement ;]
[PROPERTYDEFINITIONS statement]
[  LAYER (Nonrouting) statement
 | LAYER (Routing) statement] ...
[SPACING statement]
[MAXVIASTACK statement]
[VIA statement] ...    #Fixed vias that can be used inside VIARULE
[VIARULE statement] ...
[VIARULE GENERATE statement] ...
[VIA statement] ...    #Generated vias that can reference VIARULE name
[NONDEFAULTRULE statement] ...
[SITE statement] ...
[MACRO statement
  [PIN statement] ...
  [OBS statement ...]] ...
[BEGINEXT statement] ...
[END LIBRARY]
```

LEF Statement Definitions

The following definitions describe the syntax arguments for the statements that make up a LEF file. Statements are listed in alphabetical order, *not* in the order they should appear in a LEF file. For the correct order, see "Order of LEF Statements" on page 13.

Bus Bit Characters

`[BUSBITCHARS "delimiterPair" ;]`

Specifies the pair of characters used to specify bus bits when LEF names are mapped to or from other databases. The characters must be enclosed in double quotation marks. For example:

```
BUSBITCHARS "[]" ;
```

If one of the bus bit characters appears in a LEF name as a regular character, you must use a backslash (\) before the character to prevent the LEF reader from interpreting the character as a bus bit delimiter.

If you do not specify the `BUSBITCHARS` statement in your LEF file, the default value is `"[]"`.

Clearance Measure

```
[CLEARANCEMEASURE {MAXXY | EUCLIDEAN} ;]
```

Defines the clearance spacing requirement that will be applied to pins and obstructions (blockages) in cells. If you do not specify a `CLEARANCEMEASURE` statement, euclidean distance is used by default.

`MAXXY`	Uses the largest x or y distances for spacing between objects.
`EUCLIDEAN`	Uses the euclidean distance for spacing between objects. That is, the square root of $x^2 + y^2$.

Divider Character

```
[DIVIDERCHAR "character" ;]
```

Specifies the character used to express hierarchy when LEF names are mapped to or from other databases. The character must be enclosed in double quotation marks. For example:

```
    DIVIDERCHAR "/" ;
```

If the divider character appears in a LEF name as a regular character, you must use a backslash (\) before the character to prevent the LEF reader from interpreting the character as a hierarchy delimiter.

If you do not specify the `DIVIDERCHAR` statement in your LEF file, the default value is *`"/"`*.

Extensions

```
[BEGINEXT "tag"
     extension
ENDEXT]
```

Adds customized syntax to the LEF file that can be ignored by tools that do not use that syntax. You can also use extensions to add new syntax not yet supported by your version of LEF/DEF, if you are using version 5.1 or later.

`extension`	Specifies the contents of the extension.
`"`*`tag`*`"`	Identifies the extension block. You must enclose *`tag`* in double quotation marks.

Example 1-1 Extension Statement

```
BEGINEXT "1VSI Signature 1.0"
    CREATOR "company name"
    DATE "timestamp"
    REVISION "revision number"
ENDEXT
```

Layer (Cut)

```
LAYER layerName
     TYPE CUT ;
     [SPACING minSpacing
         [ LAYER 2ndLayerName
         | ADJACENTCUTS {3 | 4} WITHIN distance
         | CENTERTOCENTER]
     ;] ...
     [WIDTH minWidth ;]
     [ENCLOSURE [ABOVE | BELOW] overhang1 overhang2 [WIDTH minWidth] ;] ...
     [PREFERENCLOSURE [ABOVE | BELOW overhang1 overhang2 [WIDTH minWidth] ;] ...
     [RESISTANCE resistancePerCut ;]
     [PROPERTY propName propVal ;] ...
     [ACCURRENTDENSITY {PEAK | AVERAGE | RMS}
       { value
       | FREQUENCY freq_1 freq_2 ... ;
           [CUTAREA cutArea_1 cutArea_2 ... ;]
           TABLEENTRIES
            v_freq_1_cutArea_1 v_freq_1_cutArea_2 ...
            v_freq_2_cutArea_1 v_freq_2_cutArea_2 ...
            ...
       } ;]
     [DCCURRENTDENSITY AVERAGE
       { value
       | CUTAREA cutArea_1 cutArea_2 ... ;
           TABLEENTRIES value_1 value_2 ...
       } ;]
     [ANTENNAMODEL {OXIDE1 | OXIDE2 | OXIDE3 | OXIDE4} ;] ...
     [ANTENNAAREARATIO value ;] ...
     [ANTENNADIFFAREARATIO {value | PWL ( ( d1 r1 ) ( d2 r2 ) ...)} ;] ...
     [ANTENNACUMAREARATIO value ;] ...
     [ANTENNACUMDIFFAREARATIO {value | PWL ( ( d1 r1 ) ( d2 r2 ) ...)} ;] ...
     [ANTENNAAREAFACTOR value [DIFFUSEONLY] ;] ...

END layerName
```

Defines cut layers in the design. Each cut layer is defined by assigning it a name and design rules. You must define cut layers separately, with their own layer statements.

You must define layers in process order from bottom to top. For example:

```
    poly       masterslice
```

```
cut01      cut
metal1     routing
cut12      cut
metal2     routing
cut23      cut
metal3     routing
```

`ACCURRENTDENSITY`	Specifies how much AC current a cut of a certain area can handle at a certain frequency. For an example using the `ACCURRENTDENSITY` syntax, see Example 1-5 on page 33.

The `ACCURRENTDENSITY` syntax is defined as follows:

```
{PEAK | AVERAGE | RMS}
{ value
| FREQUENCY freq_1 freq_2 ... ;
   [CUTAREA cutArea_1 cutArea_2 ... ;]
   TABLEENTRIES
    v_freq_1_cutArea_1 v_freq_1_cutArea_2 ...
    v_freq_2_cutArea_1 v_freq_2_cutArea_2 ...
    ...
} ;
```

`PEAK`	Specifies the peak limit of the layer.
`AVERAGE`	Specifies the average limit of the layer.
`RMS`	Specifies the root mean square limit of the layer.
`value`	Specifies a maximum current limit for the layer in milliamps per square micron (mA/μm^2). *Type:* Float
`FREQUENCY`	Specifies frequency values, in megahertz. You can specify more than one frequency. If you specify multiple frequency values, the values must be specified in ascending order. If you specify only one frequency value, there is no frequency dependency, and the table entries are assumed to apply to all frequencies. *Type:* Float

`CUTAREA`	Specifies cut area values, in square microns (µm^2). You can specify more than one cut area. If you specify multiple cut area values, the values must be specified in ascending order. If you specify only one cut area value, there is no cut area dependency, and the table entries are assumed to apply to all cut areas. *Type:* Float
`TABLEENTRIES`	Defines the maximum current for each frequency and cut area pair specified in the `FREQUENCY` and `CUTAREA` statements, in mA/µm^2. The pairings define each cut area for the first frequency in the `FREQUENCY` statement, then the cut areas for the second frequency, and so on. The final value for a given cut area and frequency is computed from a linear interpolation of the table values. *Type:* Float

`ANTENNAAREAFACTOR` *`value`* `[DIFFUSEONLY]`

Specifies the multiply factor for the antenna metal area calculation. `DIFFUSEONLY` specifies that the current antenna factor should only be used when the corresponding layer is connected to the diffusion.
Default: `1.0`
Type: Float

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

Note: If you specify a value that is greater than 1.0, the computed areas will be larger, and violations will occur more frequently.

`ANTENNAAREARATIO` *`value`*

Specifies the maximum legal antenna ratio, using the area of the

metal wire that is not connected to the diffusion diode. For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"
Type: Integer

`ANTENNACUMAREARATIO` *`value`*

Specifies the cumulative antenna ratio, using the area of the metal wire that is not connected to the diffusion diode. For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"
Type: Integer

`ANTENNACUMDIFFAREARATIO {`*`value`*` | PWL ( ( `*`d1 r1`*` ) ( `*`d2 r2`*` )...)}`

Specifies the cumulative antenna ratio, using the area of the metal wire that is connected to the diffusion diode. You can supply an explicit ratio *`value`* or specify piece-wise linear format (`PWL`), in which case the cumulative ratio is calculated using linear interpolation of the diffusion area and ratio input values. The diffusion input values must be specified in ascending order.
Type: Integer

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNADIFFAREARATIO {`*`value`*` | PWL ( ( `*`d1 r1`*` ) ( `*`d2 r2`*` )...)}`

Specifies the antenna ratio, using the area of the metal wire connected to the diffusion diode. You can supply an explicit ratio *`value`* or specify piece-wise linear format (`PWL`), in which case the ratio is calculated using linear interpolation of the diffusion area and ratio input values. The diffusion input values must be specified in ascending order.
Type: Integer

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAMODEL {OXIDE1 | OXIDE2 | OXIDE3 | OXIDE4}`

Specifies the oxide model for the layer. If you specify an `ANTENNAMODEL` statement, that value affects all `ANTENNA*` statements for the layer that follow it until you specify another

	ANTENNAMODEL statement. *Default:* OXIDE1, for a new LAYER statement Because LEF is sometimes used incrementally, if an ANTENNA statement occurs twice for the same oxide model, the last value specified is used. For any given ANTENNA keyword, only one value or PWL table is stored for each oxide metal on a given layer. For an example using the ANTENNAMODEL syntax, see Example 1-6 on page 37.
CENTERTOCENTER	Specifies that the minimum spacing rule applies from the center-to-center of two cuts, instead of from the edge-to-edge.
DCCURRENTDENSITY	Specifies how much DC current a via cut of a certain area can handle in units of milliamps per square micron (mA/μm^2). For an example using the DCCURRENTDENSITY syntax, see Example 1-7 on page 39.

The DCCURRENTDENSITY syntax is defined as follows:

```
AVERAGE
{ value
| CUTAREA cutArea_1 cutArea_2 ... ;
   TABLEENTRIES value_1 value_2 ...
};
```

AVERAGE	Specifies the average limit for the layer.
value	Specifies a current limit for the layer in mA/μm^2. *Type:* Float
CUTAREA	Specifies cut area values, in square microns. You can specify more than one cut area value. If you specify multiple cut area values, the values must be specified in ascending order. *Type:* Float

`TABLEENTRIES` Specifies the maximum current density for each specified cut area, in mA/µm^2. The final value for a specific cut area is computed from a linear interpolation of the table values.
Type: Float

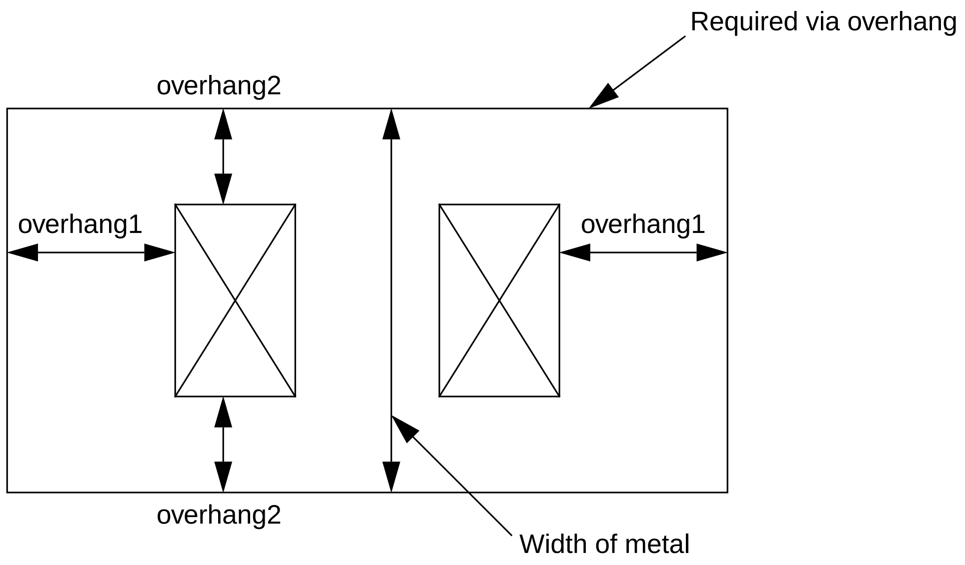
`ENCLOSURE [ABOVE | BELOW]` *`overhang1 overhang2`* `[WIDTH` *`minWidth`*`]`

Specifies that any rectangle from this cut layer requires the routing layers to overhang by *`overhang1`* on two opposite sides, and by *`overhang2`* on the other two opposite sides. (See Figure 1-1 on page 22.)

If you specify `BELOW`, the overhang is required on the routing layers below this cut layer. If you specify `ABOVE`, the overhang is required on the routing layers above this cut layer. If you specify neither, the rule applies to both adjacent routing layers.

If you specify a `WIDTH` value, then the enclosure rule only applies when the width of the routing layer is greater than or equal to *`minWidth`*. If you do not specify a minimum width, the enclosure rule applies to all widths. If you specify multiple enclosure rules with different minimum width values, the largest *`minWidth`* rule that is still less than or equal to the wire width applies. For example, if you specify enclosure rules for 1.0 µm and 2.0 µm widths, a 1.5 µm wire uses the 1.0 rule, and a 2.0 µm wire uses the 2.0 rule. See Example 1-2 on page 22.
Type: Float, specified in microns, for all values

Figure 1-1 Enclosure Rule



Example 1-2 Enclosure Statements

The following definition describes a cut layer that has a minimum center-to-center spacing of 0.15 µm, and includes `PREFERENCLOSURE` rules:

```
LEF via12
    TYPE CUT ;
    WIDTH 0.20 ;                     #cuts .20 x .20 squares
    SPACING 0.15 CENTERTOCENTER ;    #via12 center-to-center spacing is 0.15
    ENCLOSURE BELOW .03 .01 ;        #m1: 0.03 on two sides, 0.01 on other sides
    ENCLOSURE ABOVE .05 .01 ;        #m2: 0.05 on two sides, 0.01 on other sides
    PREFERENCLOSURE BELOW 0.06 0.01 ; #Prefer m1: 0.06 on two sides, 0.01 on others
    PREFERENCLOSURE ABOVE 0.08 0.02 ; #Prefer m2: 0.08 on two sides, 0.02 on others
    RESISTANCE 10.0 ;                #10.0 ohms per cut
    ...
END via12
```

The following definition describes a cut layer that has a minimum edge-to-edge spacing of 0.15 µm:

```
LAYER via23
    TYPE CUT ;
    WIDTH 0.20                   #cuts .20 x .20 squares
    SPACING 0.15 ;               #via23 edge-to-edge spacing is 0.15
    ENCLOSURE .05 .01 ;          #m2, m3: 0.05 on two sides, 0.01 on other sides
    ENCLOSURE .02 .02 WIDTH 1.0 ; #m2 needs 0.02 on all sides if m2 width >=1.0
```

```
                             #m3 needs 0.02 on all sides if m3 width >=1.0
    ENCLOSURE .05 .05 WIDTH 2.0 ; #m2 needs 0.05 on all sides if m2 width >=2.0
                             #m3 needs 0.05 on all sides if m3 width >=2.0
    RESISTANCE 10.0 ;         #10.0 ohms per cut
    ...
END via23
```

The following definition describes a cut layer that requires an overhang of .07 µm on all sides of *metal3* and an overhang of .09 µm on all sides of *metal4*, if the widths of *metal3* and *metal4* are greater than or equal to 1.0 µm:

```
LAYER via34
    TYPE CUT ;
    WIDTH 0.25 ;                        #cuts .25 x .25 squares
    ENCLOSURE .05 .01 ;                 #m3, m4 must meet enclosure rule
    ENCLOSURE BELOW .07 .07 WIDTH 1.0 ; #m3 needs .07 on all sides if the m3
                                        #width is >= 1.0
    ENCLOSURE ABOVE .09 .09 WIDTH 1.0 ; #m4 needs .09 on all sides if the m4
                                        #width is >= 1.0
    RESISTANCE 8.0 ;                    #8.0 ohms per cut
    ...
END via34
```

LAYER *LayerName*
Specifies the name for the layer. This name is used in later references to the layer.

PREFERENCLOSURE [ABOVE | BELOW] *overhang1 overhang2* [WIDTH *minWidth*]
Specifies preferred enclosure rules that can improve manufacturing yield, instead of enclosure rules that absolutely must be met (see the ENCLOSURE keyword). Applications should use the PREFERENCLOSURE rule when it has little or no impact on density and routability.

PROPERTY *propName propVal*
Specifies a numerical or string value for a layer property defined in the PROPERTYDEFINITIONS statement. The *propName* you specify must match the *propName* listed in the PROPERTYDEFINITIONS statement.

RESISTANCE *resistancePerCut*
Specifies the resistance per cut on this layer. LEF vias without their own specific resistance value, or DEF vias from a VIARULE without a resistance per cut value, can use this resistance value.

Via resistance is computed using *`resistancePerCut`* and Kirchoff's law for typical parallel resistance calculation. For example, if R =10 ohms per cut, and the via has one cut, then R =10 ohms. If the via has two cuts, then R = (1/2) * 10 = 5 ohms.

`SPACING`

Specifies the minimum spacing allowed between via cuts on the same net or different nets. For via cuts on the same net, this value can be overridden by the `SAMENET SPACING` statement. For an example using the `SPACING` syntax, see Example 1-3 on page 25.

The `SPACING` syntax is defined as follows:

```
[SPACING minSpacing
    [ LAYER 2ndLayerName
    | ADJACENTCUTS {3 | 4} WITHIN distance]
;] ...
```

`minSpacing`

Specifies the default minimum spacing between via cuts, in microns.
Type: Float

`LAYER` *`2ndLayerName`*

Applies the spacing rule between objects on the cut layer and objects on *`2ndLayerName`*. The second cut layer must already be defined in the LEF file.

`ADJACENTCUTS {3 | 4} WITHIN` *`distance`*

Applies the spacing rule only when the cut has at least three or four via cuts that are less than *`distance`*, in microns, from each other. You can specify only one `ADJACENTCUTS` statement per cut layer. For more information, see "Adjacent Via Cuts" on page 25.
Type: Float (*`distance`*)

`TYPE CUT`

Specifies that the layer is for contact-cuts. The type is later referenced in vias and in rules for generating vias.

`WIDTH` *`minWidth`*

Specifies the minimum width of a cut. In most technologies, this is also the only legal size of a cut.
Type: Float, specified in microns

Example 1-3 Spacing Rule

The following SPACING rule specifies that extra space is needed for any via with more than three adjacent cuts, which happens if one via has more than 2x2 cuts. A cut that is within .25 µm of three other cuts requires spacing that is greater than or equal to 0.22 µm.

```
LAYER CUT12
    SPACING 0.20 ;                          #default cut spacing
    SPACING 0.22 ADJACENTCUTS 3 WITHIN 0.25 ;
    ...
END CUT12
```

The following SPACING rule specifies that extra space is required for any via with 3x3 cuts or more (that is, a cut with four or more adjacent cuts). A cut that is within .25 µm of four other cuts requires spacing that is greater than or equal to 0.22 µm.

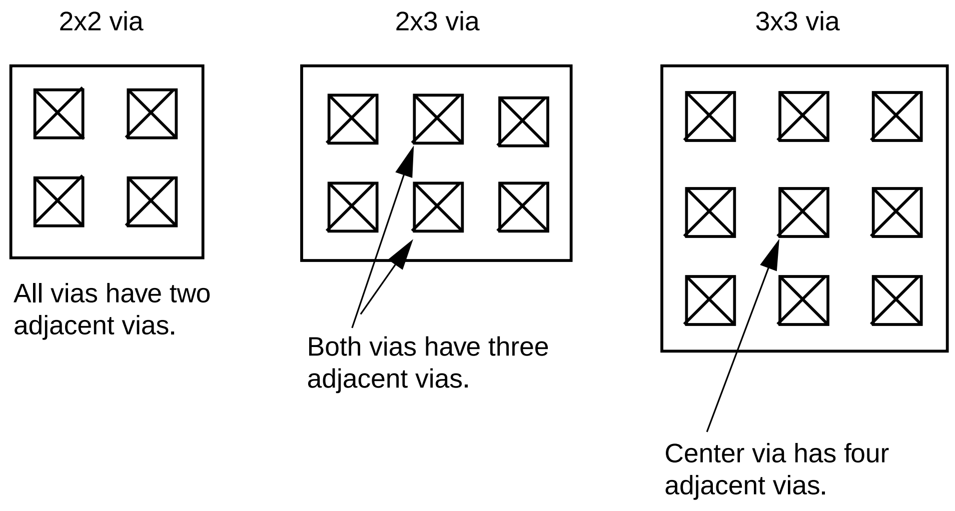
```
LAYER CUT12
    SPACING 0.20 ;                          #default cut spacing
    SPACING 0.22 ADJACENTCUTS 4 WITHIN 0.25 ;
    ...
END CUT12
```

Adjacent Via Cuts

A cut is considered adjacent if it is within *distance* of another cut in any direction (including a 45-degree angle). Figure 1-2 on page 26 illustrates adjacent via cuts for 2x2, 2x3, and 3x3 vias, for typical spacing values (that is, the diagonal spacing is greater than the ADJACENTCUTS distance value). For three adjacent cuts, the ADJACENTCUTS rule allows tight cut spacing on 1x*n* vias and 2x2 vias, but requires larger cut spacing on 2x3, 2x4 and 3x*n* vias. For four adjacent cuts, the rule allows tight cut spacing on 2x*n* vias, but it requires larger cut spacing on 3x*n* vias.

The ADJACENTCUTS rule overrides the cut-to-cut spacing used in VIARULE GENERATE statements for large vias if the ADJACENTCUTS spacing value is larger that the VIARULE spacing value.

Figure 1-2


2x2 via
2x3 via
3x3 via
All vias have two adjacent vias.
Both vias have three adjacent vias.
Center via has four adjacent vias.

Layer (Implant)

```
LAYER layerName
     TYPE IMPLANT ;
     [WIDTH minWidth ;]
     [SPACING minSpacing [LAYER layerName2] ;] ...
     [PROPERTY propName propVal ;] ...
END layerName
```

Defines implant layers in the design. Each layer is defined by assigning it a name and simple spacing and width rules. These spacing and width rules only affect the legal cell placements. These rules interact with the library methodology, detailed placement, and filler cell support. You must define implant layers separately, with their own layer statements.

`LAYER` *`layerName`*	Specifies the name for the layer. This name is used in later references to the layer.
`LAYER` *`layerName2`*	Specifies the name of another implant layer that requires extra spacing that is greater than or equal to *`minspacing`* from this implant layer.

`PROPERTY` *`propName propVal`*

Specifies a numerical or string value for a layer property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`SPACING` *`minSpacing`*	Specifies the minimum spacing for the layer. This value affects the legal cell placement. *Type:* Float, specified in microns
`TYPE IMPLANT`	Identifies the layer as an implant layer.
`WIDTH` *`minWidth`*	Specifies the minimum width for this layer. This value affects the legal cell placement. *Type:* Float, specified in microns

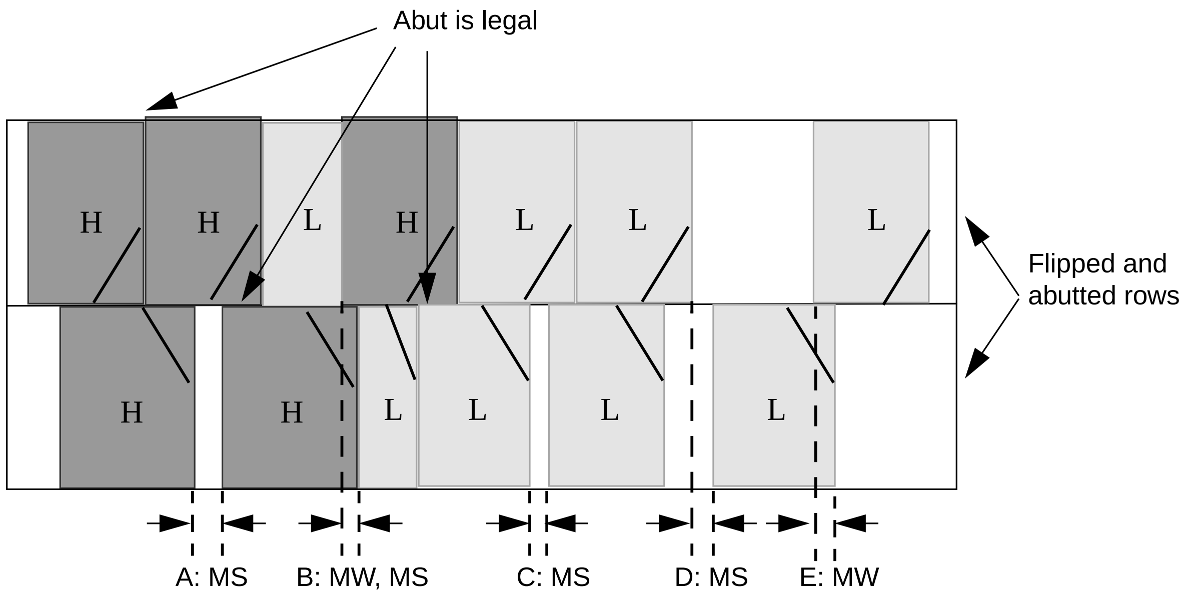
Example 1-4 Implant Layer

Typically, you define high-drive cells on one implant layer and low-drive cells on another implant layer. The following example defines high-drive cells on *implant1* and low-drive cells on *implant2*. Both implant layers cover the entire cell. The placer and filler cell creation attempt to legalize the cell overlaps in abutting rows to ensure that the minimum width and spacing values are met.

```
LAYER implant1          #high-drive implant layer
    TYPE IMPLANT ;
    WIDTH 0.50 ;        #implant rectangles must be >=0.50 microns wide
    SPACING 0.50 ;      #implant rectangles must be >=0.50 microns apart
END implant1

LAYER implant2          #low-drive imlpant layer
       TYPE IMPLANT ;
       WIDTH 0.50 ;     #implant rectangles must be >=0.50 microns wide
       SPACING 0.50 ;   #implant rectangles must be >=0.50 microns apart
END implant2
```

Assume that the high-drive cells and low-drive cells are completely covered by their respective implant layers. Because there is no spacing between *implant1* and *implant2* specified, you might see a placement like that illustrated in Figure 1-3 on page 28.

Figure 1-3



MS = Minimum spacing error
MW = Minimum width error

Note that you can correct A, C, D, and E by putting in filler cells with the appropriate implant type. However, B cannot be corrected by a filler cell—either the placer must avoid it, or you must allow the filler cell or post-process command to move cells or modify the implant layer to correct the error.

Layer (Masterslice or Overlap)

```
LAYER layerName
     TYPE {MASTERSLICE | OVERLAP} ;
     [PROPERTY propName propVal ;] ...

END layerName
```

Defines masterslice (nonrouting) or overlap layers in the design. Masterslice layers are typically polysilicon layers and are only needed if the cell `MACRO`s have pins on the polysilicon layer.

The overlap layer should normally be named `OVERLAP`. It can be used in `MACRO` definitions to form rectilinear-shaped cells and blocks (that is, an “L”-shaped block).

Each layer is defined by assigning it a name and design rules. You must define masterslice or overlap layers separately, with their own layer statements.

You must define layers in process order from bottom to top. For example:

```
poly        masterslice
cut01       cut
metal1      routing
cut12       cut
metal2      routing
cut23       cut
metal3      routing
```

`LAYER` *`layerName`*	Specifies the name for the layer. This name is used in later references to the layer.
`TYPE`	Specifies the purpose of the layer.

`MASTERSLICE`	Layer is fixed in the base array. If pins appear in the masterslice layers, you must define vias to permit the routers to connect those pins and the first routing layer. Wires are not allowed on masterslice layers. Routing tools can use only one masterslice layer. If a masterslice layer is defined, exactly one cut layer must be defined between the masterslice layer and the adjacent routing layers.
`OVERLAP`	Layer used for overlap checking for rectilinear blocks. Obstruction descriptions in the macro obstruction statements refer to the overlap layer.

`PROPERTY` *`propName propVal`*

Specifies a numerical or string value for a layer property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

Layer (Routing)

```
LAYER layerName
     TYPE ROUTING ;
     DIRECTION {HORIZONTAL | VERTICAL | DIAG45 | DIAG135} ;
     PITCH {distance | xDistance yDistance} ;
     [DIAGPITCH {distance | diag45Distance diag135Distance} ;]
     WIDTH defaultWidth ;
```

```
[OFFSET {distance | xDistance yDistance} ;]
[DIAGWIDTH diagWidth ;]
[DIAGSPACING diagSpacing ;]
[DIAGMINEDGELENGTH diagLength ;]
[AREA minArea ;]
[MINSIZE minWidth minLength [minWidth2 minLength2] ... ;]
[ [SPACING minSpacing
    [ RANGE minWidth maxWidth
       [ USELENGTHTHRESHOLD
       | INFLUENCE value [RANGE stubMinWidth stubMaxWidth]
       | RANGE minWidth maxWidth]
    | LENGTHTHRESHOLD maxLength [RANGE minWidth maxWidth]
    ]
  ;] ...
| [SPACINGTABLE
    PARALLELRUNLENGTH {length} ...
        {WIDTH width {spacing} ...} ... ;
    [SPACINGTABLE
      INFLUENCE {WIDTH width WITHIN distance SPACING spacing} ... ;
  ] ]
]
[WIREEXTENSION value ; ]
[MINIMUMCUT numCuts WIDTH width
   [FROMABOVE | FROMBELOW]
   [LENGTH length WITHIN distance] ;] ...
[MAXWIDTH width ;]
[MINWIDTH width ;]
[MINSTEP minStepLength
   [INSIDECORNER | OUTSIDECORNER | STEP]
   [LENGTHSUM maxLength] ;]
[MINENCLOSEDAREA area [WIDTH width] ;] ...
[PROTRUSIONWIDTH width1 LENGTH length WIDTH width2 ;]
[RESISTANCE RPERSQ value ;]
[CAPACITANCE CPERSQDIST value ;]
[HEIGHT distance ;]
[THICKNESS distance ;]
[SHRINKAGE distance ;]
[CAPMULTIPLIER value ;]
[EDGECAPACITANCE value ;]
[SLOTWIREWIDTH minWidth ;]
[SLOTWIRELENGTH minLength ;]
[SLOTWIDTH minWidth ;]
[SLOTLENGTH minLength ;]
[MAXADJACENTSLOTSPACING spacing ;]
[MAXCOAXIALSLOTSPACING spacing ;]
[MAXEDGESLOTSPACING spacing ;]
[SPLITWIREWIDTH minWidth ;]
[MINIMUMDENSITY minDensity ;]
[MAXIMUMDENSITY maxDensity ;]
[DENSITYCHECKWINDOW windowLength windowWidth ;]
[DENSITYCHECKSTEP stepValue ;]
```

```
     [FILLACTIVESPACING spacing ;]
     [ANTENNAMODEL {OXIDE1 | OXIDE2 | OXIDE3 | OXIDE4} ;] ...
     [ANTENNAAREARATIO value ;] ...
     [ANTENNADIFFAREARATIO {value | PWL ( ( d1 r1 ) ( d2 r2 ) ...) } ;] ...
     [ANTENNACUMAREARATIO value ;] ...
     [ANTENNACUMDIFFAREARATIO {value | PWL ( ( d1 r1 ) ( d2 r2 ) ...) } ;] ...
     [ANTENNAAREAFACTOR value [DIFFUSEONLY] ;] ...
     [ANTENNASIDEAREARATIO value ;] ...
     [ANTENNADIFFSIDEAREARATIO {value | PWL ( ( d1 r1 ) ( d2 r2 ) ...) } ;] ...
     [ANTENNACUMSIDEAREARATIO value ;] ...
     [ANTENNACUMDIFFSIDEAREARATIO {value | PWL ( ( d1 r1 ) ( d2 r2 ) ...) } ;] ...
     [ANTENNASIDEAREAFACTOR value [DIFFUSEONLY] ;] ...
     [PROPERTY propName propVal ;] ...
     [ACCURRENTDENSITY {PEAK | AVERAGE | RMS}
        { value
        | FREQUENCY freq_1 freq_2 ... ;
            [WIDTH width_1 width_2 ... ;]
            TABLEENTRIES
              v_freq_1_width_1 v_freq_1_width_2 ...
              v_freq_2_width_1 v_freq_2_width_2 ...
              ...
        } ;]
     [DCCURRENTDENSITY AVERAGE
        { value
        | WIDTH width_1 width_2 ... ;
            TABLEENTRIES value_1 value_2 ...
        } ;]
END layerName
```

Defines routing layers in the design. Each layer is defined by assigning it a name and design rules. You must define routing layers separately, with their own layer statements.

You must define layers in process order from bottom to top. For example:

```
    poly       masterslice
    cut01      cut
    metal1     routing
    cut12      cut
    metal2     routing
    cut23      cut
    metal3     routing
```

`ACCURRENTDENSITY` Specifies how much AC current a wire on this layer of a certain width can handle at a certain frequency in units of milliamps per micron (mA/µm).

Note: The true meaning of current density would have units of milliamps per square micron (mA/µm^2); however, the thickness of the metal layer is implicitly included, so the units in this table are milliamps per micron, where only the wire width varies.

The `ACCURRENTDENSITY` syntax is defined as follows:

```
{PEAK | AVERAGE | RMS}
{ value
| FREQUENCY freq_1 freq_2 ... ;
   [WIDTH width_1 width_2 ... ;]
   TABLEENTRIES
     v_freq_1_width_1 v_freq_1_width_2 ...
     v_freq_2_width_1 v_freq_2_width_2 ...
     ...
} ;
```

`PEAK`	Specifies the peak current limit of the layer.
`AVERAGE`	Specifies the average current limit of the layer.
`RMS`	Specifies the root mean square current limit of the layer.
`value`	Specifies a maximum current for the layer in mA/µm. *Type:* Float
`FREQUENCY`	Specifies frequency values, in megahertz. You can specify more than one frequency. If you specify multiple frequency values, the values must be specified in ascending order. If you specify only one frequency value, there is no frequency dependency, and the table entries are assumed to apply to all frequencies. *Type:* Float

`WIDTH`	Specifies wire width values, in microns. You can specify more than one wire width. If you specify multiple width values, the values must be specified in ascending order. If you specify only one width value, there is no width dependency, and the table entries are assumed to apply to all widths. *Type:* Float
`TABLEENTRIES`	Defines the maximum current for each of the frequency and width pairs specified in the `FREQUENCY` and `WIDTH` statements, in mA/µm. The pairings define each width for the first frequency in the `FREQUENCY` statement, then the widths for the second frequency, and so on. The final value for a given wire width and frequency is computed from a linear interpolation of the table values. the widths are not adjusted for any process shrinkage, so the should be correct for the “drawn width”. *Type:* Float

Example 1-5 AC Current Density Statements

The following examples define AC current density tables:

```
LAYER met1
    ...
    ACCURRENTDENSITY PEAK            #peak AC current limit for met1
    FREQUENCY 100 400 ;              #2 freq values in MHz
    WIDTH
     0.4 0.8 1.6 5.0 10.0 ;          #5 width values in microns
    TABLEENTRIES
    9.0 7.5 6.5 5.4 4.7              #mA/um for 5 widths and freq_1 (100 Mhz)
    7.5 6.8 6.0 4.8 4.0              #mA/um for 5 widths and freq_2 (400 Mhz)
    ACCURRENTDENSITY AVERAGE         #avg. AC current limit for met1
    50.0 ;                           #mA/um for any width at any frequency
    ACCURRENTDENSITY RMS             #RMS AC current limit for met1
    FREQUENCY 1 ;                   #1 freq (required by syntax; not really used
    WIDTH
     0.4 0.8 1.6 5.0 20.0 ;         #5 width values in microns
    TABLEENTRIES
    7.5 6.8 6.0 4.8 4.0 ;           #mA/um for 5 widths
    ...
END met1 ;
```

The PEAK current density at .4 µm, 100 MHz is 9.0 mA/µm. Therefore, a 0.4 µm wide wire can carry 9.0 x .4 = 3.6 mA of PEAK current.

The RMS current density at .4 µm is 7.5 mA/µm. Therefore, a 0.4 µm wide wire can carry 7.5 x .4 = 3.0 mA of RMS current.

```
LAYER cut12
    ...
    ACCURRENTDENSITY PEAK       #peak AC current limit for one cut
    FREQUENCY 10 200 ;          #2 freq values in MHz
      CUTAREA 0.16 0.32         #2 cut areas in um squared
    TABLEENTRIES
    0.5 0.4 ;                   #mA/um squared for 2 cut areas at freq_1 (10 Mhz)
    0.4 0.35 ;                  #mA/um squared for 2 cut areas at freq_2 (200 Mhz)
    ACCURRENTDENSITY AVERAGE    #average AC current limit for via cut12
    10.0 ;                      #mA/um squared for any cut area at any frequency
    ACCURRENTDENSITY RMS        #RMS AC current limit for via cut12
    FREQUENCY 1 ;               #1 freq (required by syntax; not really used)
      CUTAREA 0.16 1.6 ;        #2 cut areas in um squared
    TABLEENTRIES
    10.0 9.0 ;                  #mA/um squared for 2 cut areas at any frequency
    ....
END cut12 ;
```

`ANTENNAAREAFACTOR` *`value`* `[DIFFUSEONLY]`

Specifies the multiply factor for the antenna metal area calculation. `DIFFUSEONLY` specifies that the current antenna factor should only be used when the corresponding layer is connected to the diffusion.
Default: `1.0`
Type: Float

For more information on process antenna calculation, see Appendix C, “Calculating and Fixing Process Antenna Violations,”

Note: If you specify a value that is greater than 1.0, the computed areas will be larger, and violations will occur more frequently.

`ANTENNAAREARATIO` *`value`*

Specifies the maximum legal antenna ratio, using the area of the metal wire that is not connected to the diffusion diode. For more information on process antenna calculation, see Appendix C, “Calculating and Fixing Process Antenna Violations,”
Type: Integer

`ANTENNACUMAREARATIO` *`value`*

Specifies the cumulative antenna ratio, using the area of the wire that is not connected to the diffusion diode. For more information on process antenna calculation, see Appendix C, “Calculating and Fixing Process Antenna Violations,”
Type: Integer

`ANTENNACUMDIFFAREARATIO {`*`value`*` | PWL ( ( `*`d1 r1`*` ) ( `*`d2 r2`*` )...)}`

Specifies the cumulative antenna ratio, using the area of the metal wire that is connected to the diffusion diode. You can supply and explicit ratio *`value`* or specify piece-wise linear format (`PWL`), in which case the cumulative ratio value is calculated using linear interpolation of the diffusion area and ratio input values. The diffusion input values must be specified in ascending order.
Type: Integer

For more information on process antenna calculation, see Appendix C, “Calculating and Fixing Process Antenna Violations,”

`ANTENNACUMDIFFSIDEAREARATIO {`*`value`*` | PWL ( ( `*`d1 r1`*` ) ( `*`d2 r2`*` )...)}`

Specifies the cumulative antenna ratio, using the side wall area of the metal wire that is connected to the diffusion diode. You can supply and explicit ratio *`value`* or specify piece-wise linear format (`PWL`), in which case the cumulative ratio value is calculated using linear interpolation of the diffusion area and ratio input values. The diffusion input values must be specified in

ascending order.
Type: Integer

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNACUMSIDEAREARATIO` *`value`*

Specifies the cumulative antenna ratio, using the side wall area of the metal wire that is not connected to the diffusion diode. For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNADIFFAREARATIO {`*`value`*` | PWL ( ( `*`d1 r1`*` ) ( `*`d2 r2`*` )...)}`

Specifies the antenna ratio, using the area of the metal wire that is connected to the diffusion diode. You can supply and explicit ratio *`value`* or specify piece-wise linear format (`PWL`), in which case the ratio value is calculated using linear interpolation of the diffusion area and ratio input values. The diffusion input values must be specified in ascending order.
Type: Integer

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNADIFFSIDEAREARATIO {`*`value`*` | PWL ( ( `*`d1 r1`*` ) ( `*`d2 r2`*` )...)}`

Specifies the antenna ratio, using the side wall area of the metal wire that is connected to the diffusion diode. You can supply and explicit ratio *`value`* or specify piece-wise linear format (`PWL`), in which case the ratio value is calculated using linear interpolation of the diffusion area and ratio input values. The diffusion input values must be specified in ascending order.
Type: Integer

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAMODEL {OXIDE1 | OXIDE2 | OXIDE3 | OXIDE4}`

Specifies the oxide model for the layer. If you specify an `ANTENNAMODEL` statement, that value affects all `ANTENNA*`

statements for the layer that follow it until you specify another ANTENNAMODEL statement.
Default: OXIDE1, for a new LAYER statement

Because LEF is sometimes used incrementally, if an ANTENNA statement occurs twice for the same oxide model, the last value specified is used. For any given ANTENNA keyword, only one value or PWL table is stored for each oxide metal on a given layer.

Example 1-6 Antenna Model Statement

The following example defines antenna information for oxide models on layer *metal1*.

```
LAYER metal1
    ANTENNAMODEL OXIDE1 ;          #OXIDE1 not required, but good practice
    ANTENNACUMAREARATIO 5000 ;     #OXIDE1 values
    ANTENNACUMDIFFAREARATIO 8000 ;
    ANTENNAMODEL OXIDE2 ;          #OXIDE2 model starts here
    ANTENNACUMAREARATIO 500 ;      #OXIDE2 values
    ANTENNACUMDIFFAREARATIO 800 ;
    ANTENNAMODEL OXIDE3 ;
    ANTENNACUMAREARATIO 300 ;
    ANTENNACUMDIFFAREARATIO 600 ;
    ...
END metal1
```

ANTENNASIDEAREAFACTOR *value* [DIFFUSEONLY]

Specifies the multiply factor for the antenna metal side wall area calculation. DIFFUSEONLY specifies that the current antenna factor should only be used when the corresponding layer is connected to the diffusion.
Default: 1.0
Type: Float

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

ANTENNASIDEAREARATIO *value*

Specifies the antenna ratio, using the side wall area of the metal

wire that is not connected to the diffusion diode. For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"
Type: Integer

`AREA` *`minArea`*

Specifies the minimum metal area required for polygons on the layer. All polygons must have an area that is greater than or equal to *`minArea`*, if no `MINSIZE` rule exists. If a `MINSIZE` rule exists, all polygons must meet either the `MINSIZE` or the `AREA` rule. For an example using these rules, see Example 1-11 on page 46.
Type: Float, specified in microns squared

`CAPACITANCE CPERSQDIST` *`value`*

Specifies the capacitance for each square unit, in picofarads per square micron. This is used to model wire-to-ground capacitance.

`CAPMULTIPLIER` *`value`*

Specifies the multiplier for interconnect capacitance to account for increases in capacitance caused by nearby wires.
Default: `1`
Type: Integer

`DCCURRENTDENSITY`

Specifies how much DC current a wire on this layer of a given width can handle in units of milliamps per micron (mA/µm).

Note: The true meaning of current density would have units of milliamps per square micron (mA/µm^2); however, the thickness of the metal layer is implicitly included, so the units in this table are milliamps per micron, where only the wire width varies.

The `DCCURRENTDENSITY` syntax is defined as follows:

```
AVERAGE
{ value
| WIDTH width_1 width_2 ... ;
    TABLEENTRIES value_1 value_2 ...
} ;
```

`AVERAGE`	Specifies the average current limit of the layer.
`value`	Specifies the current limit for the layer, in mA/µm.

`WIDTH`	Specifies wire width values, in microns. You can specify more than one wire width. If you specify multiple width values, the values must be specified in ascending order. *Type:* Float
`TABLEENTRIES`	Specifies the value of current density for each specified width, in mA/µm. The final value for a given wire width is computed from a linear interpolation of the table values. The widths are not adjusted for any process shrinkage, so they should be correct for the “drawn width”. *Type:* Float

Example 1-7 DC Current Density Statements

The following examples define DC current density tables:

```
LAYER met1
    ...
    DCCURRENTDENSITY AVERAGE    #avg. DC current limit for met1
    50.0 ;                      #mA/um for any width
```

(or)

```
    DCCURRENTDENSITY AVERAGE    #avg. DC current limit for met1
    WIDTH
      0.4 0.8 1.6 5.0 20.0      #5 width values in microns
    TABLEENTRIES
    7.5 6.8 6.0 4.8 4.0 ;       #mA/um for 5 widths
    ...
END met1 ;
```

The AVERAGE current density at 0.4 µm is 7.5 mA/µm. Therefore, a 0.4 µm wide wire can carry 7.5 x .4 = 3.0 mA of AVERAGE DC current.

```
LAYER cut12
    ...
    DCCURRENTDENSITY AVERAGE    #avg. DC current limit for via cut12
    10.0 ;                      #mA/um squared for any cut area
```

(or)

```
    DCCURRENTDENSITY AVERAGE    #avg. DC current limit for via cut12
```

```
      CUTAREA 0.16 0.32         #2 cut areas in µm2
    TABLEENTRIES
    10.0 9.0 ;                  #mA/um squared for 2 cut areas
    ...
END cut12 ;
```

`DENSITYCHECKSTEP` *`stepValue`*

Specifies the stepping distance for metal density checks, in distance units.
Type: Float

`DENSITYCHECKWINDOW` *`windowLength windowWidth`*

Specifies the dimensions of the check window, in distance units.
Type: Float

`DIAGMINEDGELENGTH` *`diagLength`*

Specifies the minimum length for a diagonal edge. Any 45-degree diagonal edge must have a length that is greater than or equal to *`diagLength`*.
Type: Float, specified in microns

`DIAGPITCH {`*`distance`*` | `*`diag45Distance diag135Distance`*`}`

Specifies the 45-degree routing pitch for the layer. Pitch is used by the router to get the best routing density.
Default: None
Type: Float, specified in microns

`distance` Specifies one pitch value that is used for both the 45-degree angle and 135-degree angle directions.

`diag45Distance diag135Distance`

Specifies the 45-degree angle pitch (the center-to-center space between 45-degree angle routes) and the 135-degree angle pitch.

`DIAGSPACING` *`diagSpacing`*

Specifies the minimum spacing allowed for a 45-degree angle shape.
Default: None
Type: Float, specified in microns

`DIAGWIDTH` *`diagWidth`*

Specifies the minimum width allowed for a 45-degree angle shape.
Default: None
Type: Float, specified in microns

`DIRECTION {HORIZONTAL | VERTICAL | DIAG45 | DIAG135}`

Specifies the preferred routing direction. Automatic routing tools attempt to route in the preferred direction on a layer. A typical case is to route horizontally on layers *metal1* and *metal3*, and vertically on layer *metal2*.

`HORIZONTAL`	Routing parallel to the x axis is preferred.
`VERTICAL`	Routing parallel to the y axis is preferred.
`DIAG45`	Routing along a 45-degree angle is preferred.
`DIAG135`	Routing along a 135-degree angle is preferred.

Note: Angles are measured counterclockwise from the positive x axis.

`EDGECAPACITANCE` *`value`*

Specifies a floating-point value of peripheral capacitance, in picofarads per micron. The place-and-route tool uses this value in two situations:

- Estimating capacitance before routing
- Calculating segment capacitance after routing

For the second calculation, the tool uses *`value`* only if you set layer thickness, or layer height, to `0`. In this situation, the peripheral capacitance is used in the following formula:

segment capacitance = (*layer capacitance per square* x *segment width* x *segment length*) + (*peripheral capacitance* x 2 (*segment width* + *segment length*))

`FILLACTIVESPACING` *`spacing`*

Specifies the spacing between metal fills and active geometries.
Type: Float

`HEIGHT` *`distance`*
Specifies the distance from the top of the ground plane to the bottom of the interconnect.
Type: Float

`LAYER` *`layerName`*
Specifies the name for the layer. This name is used in later references to the layer.

`MAXADJACENTSLOTSPACING` *`spacing`*
Specifies the maximum spacing, in distance units, allowed between two adjacent slot sections.
Type: Float

`MAXCOAXIALSLOTSPACING` *`spacing`*
Specifies the maximum spacing, in distance units, allowed between two slots in the same slot section.
Type: Float

`MAXEDGESLOTSPACING` *`spacing`*
Specifies the maximum spacing, in distance units, allowed between slot edges.
Type: Float

`MAXIMUMDENSITY` *`maxDensity`*
Specifies the maximum metal density allowed for the layer, as a percentage. The *`minDensity`* and *`maxDensity`* values represent the metal density range within which all areas of the design must fall. The metal density must be greater than or equal to *`minDensity`* and less than or equal to *`maxDensity`*.
Type: Float

Example 1-8 Minimum and Maximum Density

The following example specifies a metal density range in which the minimum metal density must be greater than or equal to 20 percent and the maximum metal density must be less than or equal to 70 percent.

```
MINIMUMDENSITY 20.0 ;
MAXIMUMDENSITY 70.0 ;
```

`MAXWIDTH` *`width`*
Specifies the maximum wire width, in microns, allowed on the layer. For example, `MAXWIDTH 10.0` specifies that the width of

every wire on the layer must be less than or equal to 10.0 µm.
Type: Float

`MINENCLOSEDAREA` *`area`* `[WIDTH` *`width`*`]`

Specifies the minimum area size limit for an empty area that is enclosed by metal (that is, a donut hole formed by the metal).

`area`	Specifies the minimum area size of the hole, in microns squared. *Type:* Float
`width`	Applies the minimum area size limit only when a hole is created from a wire that has a width that is greater than *`width`*, in microns. If any of the wires that surround the donut hole are larger than this value, the rule applies. *Type:* Float

Example 1-9 Min Enclosed Area Statement

The following `MINENCLOSEDAREA` example specifies that a hole area must be greater than or equal to 0.40 µm^2.

```
LAYER m1
    ...
    MINENCLOSEDAREA 0.40 ;
```

The following `MINENCLOSEDAREA` example specifies that a hole area must be greater than or equal to 0.30 µm^2. However, if any of the wires enclosing the hole have a width that is greater than 0.15 µm, then the hole area must be greater than or equal to 0.40 µm^2. If any of the wires enclosing the hole are larger than 0.50 µm, then the hole area must be greater than or equal to 0.80 µm^2.

```
LAYER m1
    ...
    MINENCLOSEDAREA 0.30 ;
    MINENCLOSEDAREA 0.40 WIDTH 0.15 ;
    MINENCLOSEDAREA 0.80 WIDTH 0.50 ;
```

`MINIMUMCUT` Specifies the number of cuts a via must have when it is on a wide wire or pin whose width is greater than *`width`*. The `MINIMUMCUT` rule applies to all vias touching this particular metal layer. You can specify more than one `MINIMUMCUT` rule per layer.

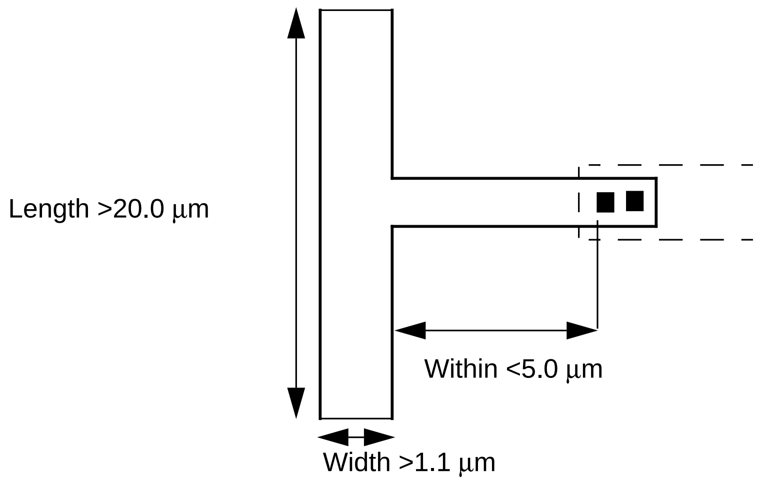
For an example using the `MINMUMCUT` syntax, see Example 1-10 on page 45.

The `MINIMUMCUT` syntax is defined as follows:

```
[MINIMUMCUT numCuts WIDTH width
  [FROMABOVE | FROMBELOW]
  [LENGTH length WITHIN distance] ] ...
```

numCuts	Specifies the number of cuts a via must have when it is on a wire or pin whose width is greater than *width*. *Type:* Integer
`WIDTH` *width*	Specifies the width of the wire or pin, in microns. *Type:* Float
`FROMABOVE | FROMBELOW`	Indicates whether the rule applies only to connections from above this layer or from below. *Default:* The rule applies to connections from above and below.
`LENGTH` *length* `WITHIN` *distance*	Indicates that the rule applies for thin wires directly connected to wide wires, if the wide wire has a width that is greater than *width* and a length that is greater than *length*, and the vias on the thin wire are less than *distance* from the wide wire (see Figure 1-4 on page 45). *Type:* Float, specified in microns

Figure 1-4



Example 1-10 Minimum Cut

The following `MINIMUMCUT` definitions show different ways to specify the `MINIMUMCUT` rule.

The following syntax specifies that two via cuts are required for *metal4* wires that are greater than 0.5 µm when connecting from *metal3* or *metal5*.

```
LAYER metal4
    MINIMUMCUT 2 WIDTH 0.5 ;
```

The following syntax specifies that four via cuts are required for *metal4* wires that are greater than 0.7 µm, when connecting from *metal3*.

```
LAYER metal4
    MINIMUMCUT 4 WIDTH 0.7 FROMBELOW ;
```

The following syntax specifies that four via cuts are required for *metal4* wires that are greater than 1.0 µm, when connecting from *metal5*.

```
LAYER metal4
    MINIMUMCUT 4 WIDTH 1.0 FROMABOVE ;
```

The following syntax specifies that two via cuts are required for *metal4* wires that are greater than 1.1 µm wide and greater than 20.0 µm long, and the via cut is less than 5.0 µm from the wide wire. Figure 1-4 on page 45 illustrates this example.

```
MINIMUMCUT 2 WIDTH 1.1 LENGTH 20.0 WITHIN 5.0 ;
```

`MINIMUMDENSITY` *`minDensity`*

Specifies the minimum metal density allowed for the layer, as a percentage. The *`minDensity`* and *`maxDensity`* values represent the metal density range within which all areas of the design must fall. The metal density must be greater than or equal to *`minDensity`* and less than or equal to *`maxDensity`*. For an example of this statement, see Example 1-8 on page 42.
Type: Float

`MINSIZE` *`minWidth minLength`* `[`*`minWidth2 minLength2`*`]`

Specifies the minimum width and length of a rectangle that must be able to fit somewhere within each polygon on this layer (see Figure 1-5 on page 47). All polygons must meet this `MINSIZE` rule, if no `AREA` rule is specified. If an `AREA` rule is specified, all polygons must meet either the `MINSIZE` or the `AREA` rule.

You can specify multiple rectangles by specifying a list of *`minWidth2`* and *`minLength2`* values. If more than one rectangle is specified, the `MINSIZE` rule is satisfied if any of the rectangles can fit within the polygon.
Type: Float, specified in microns, for all values

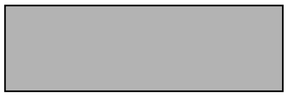
Example 1-11 Minimum Size and Area Rules

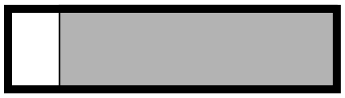
Assume the following minimum size and area rules:

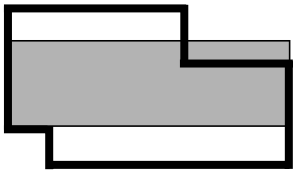
```
LAYER metal1
    TYPE ROUTING ;
    AREA 0.07 ;           #0.20 um x 0.35 um = 0.07 um^2
    MINSIZE 0.14 0.30 ;   #0.14 um x 0.30 um = 0.042 um^2
    ....
```

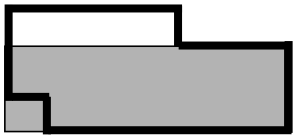
Figure 1-5 on page 47 illustrates how these rules behave when one or both of the rules are present in the `LAYER` statement:

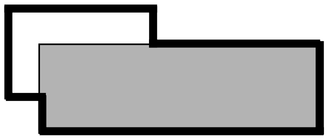
Figure 1-5 Minimum Size and Area Rules

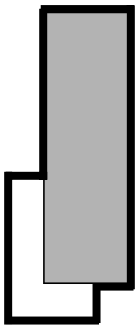
	 `MINSIZE` rule	minLength = 0.3 µm
		minWidth = 0.14 µm
		(area = 0.045 µm^2)
			AREA rule = 0.07 µm^2

a)		x = 0.4 µm
		y = 0.14 µm
		area = 0.056 µm^2
			AREA only: violation
			`MINSIZE` only: legal
			AREA and `MINSIZE`: legal

b)		x = 0.3 µm
		y = 0.26 µm
		area = 0.072 µm^2
			AREA only: legal
			`MINSIZE` only: violation
			AREA and `MINSIZE`: legal

c)		x = 0.3 µm
		y = 0.2 µm
		area = 0.054 µm^2
			AREA only: violation
			`MINSIZE` only: violation
			AREA and `MINSIZE`: violation

d)		x = 0.32 µm
		y = 0.2 µm
		area = 0.056 µm^2
			AREA only: violation
			`MINSIZE` only: legal
			AREA and `MINSIZE`: legal

e)		x = 0.2 µm
		y = 0.32 µm
		area = 0.056 µm^2
			AREA only: violation
			`MINSIZE` only: legal
			AREA and `MINSIZE`: legal

The following statement defines a `MINSIZE` rule that specifies that every polygon must have a minimum area of 0.07 µm^2, or that a rectangle of 0.14 x 0.30 µm must be able to fit within the polygon, or that a rectangle of 0.16 x 0.26 µm must be able to fit within the polygon:

```
LAYER metal1
```

```
    TYPE ROUTING ;
    AREA 0.07 ;                     #0.20 x 0.35 um = 0.07 um^2
    MINSIZE 0.14 0.30 0.16 0.26 ;  #0.14 x 0.30 um = 0.042 um^2
                                    #0.16 x 0.26 um = 0.0416 um^2
    ...
END metal1
```

`MINSTEP`	Specifies the minimum step size, or shortest edge length, for a shape. The `MINSTEP` rule ensures that Optical Pattern Correction (OPC) can be used to make the masks for this shape. As long as the total length of the edges is small enough, the OPC can succeed. For an illustration of the `MINSTEP` rules, see Figure 1-6 on page 50. For an example, see Example 1-12 on page 49. The syntax for `MINSTEP` is as follows:

```
[MINSTEP minStepLength
   [INSIDECORNER | OUTSIDECORNER | STEP]
   [LENGTHSUM maxLength] ;] ...
```

`INSIDECORNER`	Indicates that the rule applies to consecutive edges of an inside corner that are less than *`minStepLength`*. A violation occurs if two or more consecutive edges are less than *`minStepLength`*. There can only be one `INSIDECORNER` rule per layer. *Default:* `OUTSIDECORNER`
`LENGTHSUM` *`maxLength`*	Specifies the maximum total edge length allowed that OPC can correct without causing new DRC violations. A violation only occurs if the total length of consecutive edges that are less than *`minStepLength`* is greater than *`maxLength`*.
`minStepLength`	Specifies the minimum step size, or shortest edge length, for a shape. *Type:* Float, specified in microns

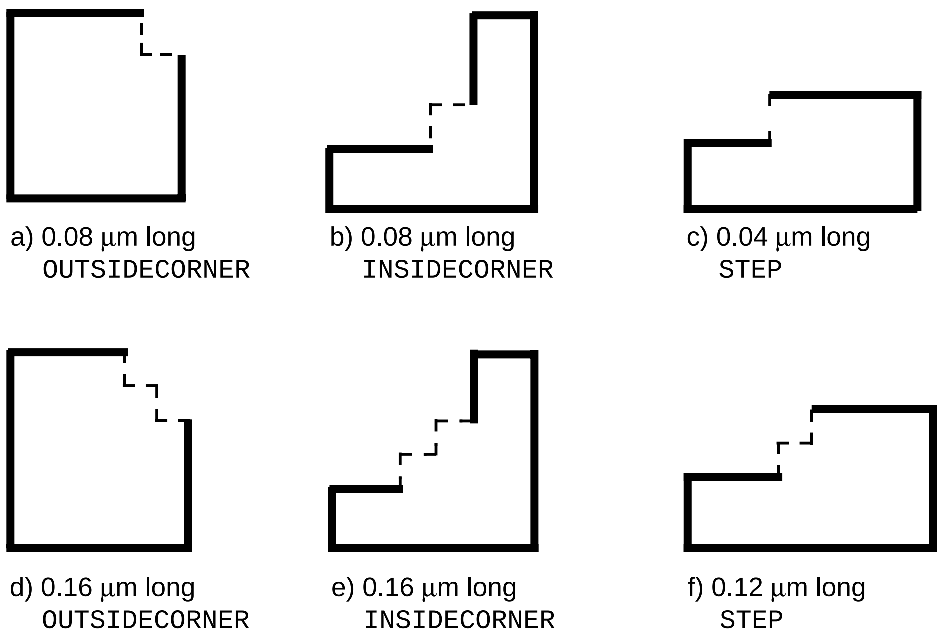
`OUTSIDECORNER`	Indicates that the rule applies to consecutive edges of an outside corner that are less than *`minStepLength`*. A DRC violation occurs if two or more consecutive edges are less than *`minStepLength`*. There can only be one `OUTSIDE` rule per layer. **Note:** This is the default rule, if `INSIDECORNER`, `OUTSIDECORNER`, or `STEP` is not specified.
`STEP`	Indicates that the rule applies to consecutive edges of a step that are less than *`minStepLength`*. A violation occurs if one or more consecutive edges are less than *`minStepLength`*. There can only be one `STEP` rule per layer. *Default:* `OUTSIDECORNER`

Example 1-12 Minimum Step Rules

Figure 1-6 on page 50 illustrates the `OUTSIDECORNER`, `INSIDECORNER`, and `STEP` rules specified in the following `MINSTEP` definitions. The definitions are shown together only for example; it is not legal to have all of them in one LEF file.

```
MINSTEP 0.05 ;                  #Previous LEF 5.5 default behavior.
                                #Therefore, OUTSIDECORNER is the default.
                                #a) and b) are violations;
                                 #b), c), e) and f) are not outside corner checks
MINSTEP 0.04 ;                   #a) and d) are legal; all edges are >= 0.04um
MINSTEP 0.05 LENGTHSUM 0.08 ;    #a) is legal, and d) is a violation
MINSTEP 0.05 LENGTHSUM 0.16 ;    #a) and d) are legal
MINSTEP 0.05 INSIDECORNER ;     #b) and e) are violations;
                                 #a), c), d), and f) are not inside corner checks
MINSTEP 0.05 INSIDECORNER LENGTHSUM 0.15 ;  #b) is legal;
                                             #e) is a LENGTHSUM violation
MINSTEP 0.05 STEP ;                #c) and f) are violations
                                    #a), b), d) and e) are not step checks
MINSTEP 0.05 STEP LENGTHSUM 0.08 ;  #c) is legal, and f) is a violation
MINSTEP 0.04 STEP ;                #c) and f) are legal;
                                    #all edges are >= 0.4 um already
```

Figure 1-6



`MINWIDTH` *`width`*

Specifies the minimum legal object width on the routing layer. For example, `MINWIDTH 0.15` specifies that the width of every object must be greater than or equal to 0.15 µm. This value is used for verification purposes, and does not affect the routing width. The `WIDTH` statement defines the default routing width on the layer.
Default: The value of the `WIDTH` statement
Type: Float, specified in microns

`OFFSET {`*`distance`*` | `*`xDistance yDistance`*`}`

Specifies the offset from the origin (`0,0`) for the routing grid for the layer.
Default: Half the routing pitch for the layer
Type: Float, specified in distance units

`distance` Specifies one offset value that is used for both the x and y offsets.

`xDistance yDistance`

Specifies the x offset for vertical routing tracks, and the y offset for horizontal routing tracks.

`PITCH {`*`distance`*` | `*`xDistance yDistance`*`}`

Specifies the required routing pitch for the layer. Pitch is used to generate the routing grid (the DEF `TRACKS`). For more information, see "Routing Pitch" on page 57.
Type: Float, specified in microns

`distance`	Specifies one pitch value that is used for both the x and y pitch.
`xDistance yDistance`	Specifies the x pitch (the space between each vertical routing track), and the y pitch (the space between each horizontal routing track).

`PROPERTY `*`propName propVal`*

Specifies a numerical or string value for a layer property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`PROTRUSIONWIDTH `*`width1`*` LENGTH `*`length`*` WIDTH `*`width2`*

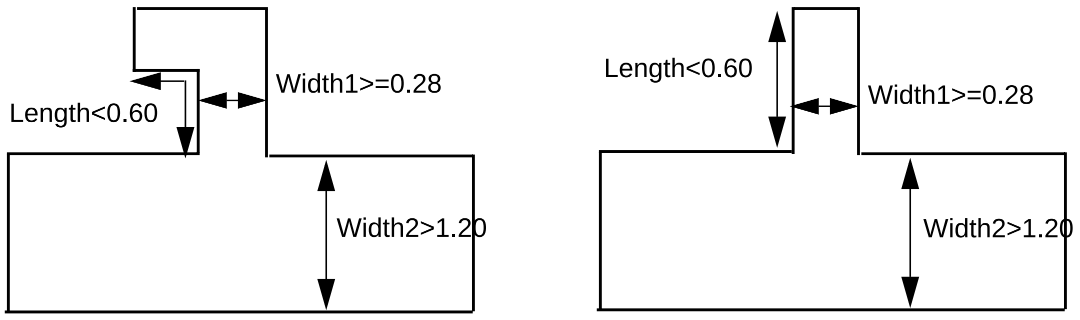
Specifies that the width of a protrusion must be greater than or equal to *`width1`* if it is shorter than *`length`*, and it connects to a wire that has a width greater than or equal to *`width2`* (see Figure 1-7 on page 52).
Type: Float, specified in microns

Example 1-13 Protrusion

The following example specifies that a protrusion must have a width that is greater than or equal to 0.28 µm, if the length of the protrusion is less than 0.60 µm and the wire it connects to has a width that is greater than 1.20 µm.

```
LAYER m1
    ...
    PROTRUSIONWIDTH 0.28 LENGTH 0.60 WIDTH 1.20 ;
    ...
```

Figure 1-7



`RESISTANCE RPERSQ` *`value`*

Specifies the resistance for a square of wire, in ohms per square. The resistance of a wire can be defined as

RPERSQU x *wire length*/*wire width*

`SHRINKAGE` *`distance`*

Specifies the value to account for shrinkage of interconnect wiring due to the etching process. Actual wire widths are determined by subtracting this constant value.
Type: Float

`SLOTLENGTH` *`minLength`*

Specifies the minimum slot length, in distance units, allowed in the design.
Default: The minimum spacing for the layer
Type: Float

`SLOTWIDTH` *`minWidth`*

Specifies the minimum slot width, in distance units, allowed in the design.
Default: The minimum spacing for the layer
Type: Float

`SLOTWIRELENGTH` *`minLength`*

Specifies the minimum wire length, in distance units, allowed for wires that need to be slotted.
Type: Float

`SLOTWIREWIDTH` *`minWidth`*

Specifies the minimum wire width, in distance units, allowed for

wires that need to be slotted.
Type: Float

`SPACING`

Specifies the spacing rules or spacing tables to use for wiring on the layer.

Note: You cannot mix `SPACING` rules and `SPACINGTABLE` rules in a `LAYER` statement. You must specify either `SPACING` statements for all routing layers or `SPACINGTABLE` statements for all routing layers.

You can specify more than one spacing rule for a layer. For information on and examples of using spacing rules, see "Using Spacing Rules" on page 57.

The syntax for describing spacing rules is defined as follows:

```
[SPACING minSpacing
  [ RANGE minWidth maxWidth
     [ USELENGTHTHRESHOLD
     | INFLUENCE influenceLength
        [RANGE stubMinWidth stubMaxWidth]
     | RANGE minWidth maxWidth]
  | LENGTHTHRESHOLD maxLength
        [RANGE minWidth maxWidth] ]
;] ...
```

`SPACING` *`minSpacing`*
`[RANGE` *`minWidth maxWidth`*`]`

Specifies the default minimum spacing, in microns, allowed between two geometries on different nets.
Type: Float

The minimum spacing rule applies to objects on the layer with widths in the indicated `RANGE` (that is, widths that are greater than or equal to *`minWidth`* and less than or equal to *`maxWidth`*). If you do not specify a range, the rule applies to all objects.
Type: Float

`USELENGTHTHRESHOLD`

Specifies that the threshold spacing rule should be used if the other object meets the previous `LENGTHTHRESHOLD` value.

`INFLUENCE` *`influenceLength`*
`[RANGE` *`stubMinWidth stubMaxWidth`*`]`

Specifies that any length of the stub wire that is less than or equal to *`influenceLength`* from the wide wire inherits the wide wire spacing.
Type: Float

The influence rule applies to stub wires on the layer with widths in the indicated `RANGE` (that is, widths that are greater than or equal to *`stubMinWidth`* and less than or equal to *`stubMaxWidth`*). If you do not specify a range, the rule applies to all stub wires.
Type: Float

Note: Specifying the `INFLUENCE` keyword denotes that the statement only checks the influence rule, and does *not* check normal spacing. You must also specify a separate `SPACING` statement for normal spacing checks.

`RANGE` *`minWidth maxWidth`*

Specifies an optional second width range. The spacing rule applies if the widths of both objects fall in the ranges defined (each object in a different range). For an object's width to fall in a range, it must be greater than or equal to *`minWidth`* and less than or equal to *`maxWidth`*.
Type: Float

`LENGTHTHRESHOLD` *`maxLength`*
`[RANGE` *`minWidth maxWidth`*`]`

Specifies the maximum parallel run length or projected length with an adjacent metal object.

The threshold spacing rule applies to objects with widths in the indicated `RANGE` (that is, widths that are greater than or equal to *`minWidth`* and less than or equal to *`maxWidth`*). If you do not specify a range, the rule applies to all objects.
Type: Float

You can specify only one parallel run length and one influence spacing table for a layer. For information on and examples of using spacing tables, see “Using Spacing Tables” on page 60.

The syntax for describing spacing tables is defined as follows:

```
[ SPACINGTABLE
    PARALLELRUNLENGTH {length} ...
       {WIDTH width {spacing} ...} ... ;
    [ SPACINGTABLE
       INFLUENCE {WIDTH width WITHIN distance
          SPACING spacing} ... ; ]
]
```

`PARALLELRUNLENGTH {`*`length`*`} ...`

Specifies the maximum parallel run length between two objects, in microns. You must specify *`length`* values in increasing order.
Type: Float

`WIDTH` *`width`* `{`*`spacing`*`} ...`

`width` specifies the maximum width of the two objects, in microns.
Type: Float

`spacing` specifies the spacing between the two wires, in microns.
Type: Float

You must specify *`width`* and *`spacing`* values in increasing order.

If the maximum width of the two objects is greater than *`width`*, and the parallel run length is greater than *`length`*, then the spacing between the objects must be greater than or equal to *`spacing`*.

`INFLUENCE` Creates a table that enforces wide wire spacing rules between nearby perpendicular wires. You can only specify an `INFLUENCE` table if you specify a `PARALLELRUNLENGTH` table first.

`WIDTH` *`width`* `WITHIN` *`distance`* `SPACING` *`spacing`* `...`

`width` specifies the width of the object, in microns.
Type: Float

`distance` specifies the distance between the object and two perpendicular wires, in microns.
Type: Float

`spacing` specifies the spacing between the two perpendicular wires, in microns.
Type: Float

You must specify *`width`* values in increasing order.

If an object has a width that is greater than *`width`*, and is located less than *`distance`* from two perpendicular wires, then the perpendicular wires must be separated by spacing that is greater than or equal to *`spacing`*.

`SPLITWIREWIDTH` *`minWidth`*	Specifies the minimum wire width, in distance units, allowed for wires that need to be split. *Type:* Float
`THICKNESS` *`distance`*	Specifies the thickness of the interconnect. *Type:* Float
`TYPE ROUTING`	Identifies the layer as a routable layer.
`WIDTH` *`defaultWidth`*	Specifies the default routing width to use for all regular wiring on the layer. *Type:* Float
`WIREEXTENSION` *`value`*	Specifies the distance by which wires are extended at vias. You must specify a value that is more than half of the routing width. *Default:* Wires are extended half of the routing width *Type:* Float

Routing Pitch

The `PITCH` statements define the detail routing grid generated when you initialize a floorplan. The pitch for a given routing layer defines the distance between routing tracks in the preferred direction for that layer. The complete routing grid is the union of the tracks generated for each routing layer.

The spacing of the grid should be no less than line-to-via spacing in both the horizontal and vertical directions. Grid spacing less than line-to-via spacing can result in routing problems and can decrease the utilization results.

The grid should normally allow for diagonal vias. Via spacing on all layers included in the via definition in LEF determines whether or not diagonal vias can be used. The router is capable of avoiding violations between diagonal vias. If you allow diagonal vias, less time is needed for routing and the layout creates a smaller design.

Using Spacing Rules

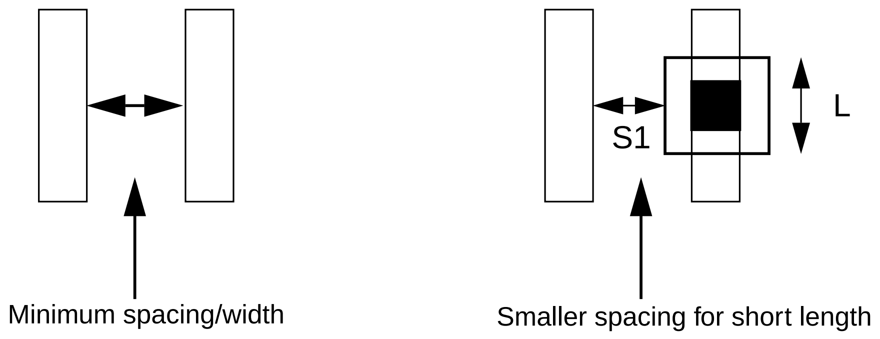
Spacing rules apply to pin-to-wire, obstruction-to-wire, via-to-wire, and wire-to-wire spacing. These requirements specify the default minimum spacing allowed between two geometries on different nets.

When defined with a `RANGE` argument, a spacing value applies to all objects with widths within a specified range. That is, the rule applies to objects whose widths are greater than or equal to the specified minimum width and less than or equal to the specified maximum width.

In the following example, the default minimum allowed spacing between two adjacent objects is 0.3 µm. However, for objects between 1.01 and 2.0 µm in width, the spacing is 0.5 µm.

```
SPACING 0.3 ;
SPACING 0.5 RANGE 1.01 2.0 ;
```

Threshold spacing is a function of both the wire width and the length of the neighboring object. It is typically used when vias are wider than the wire to allow tighter wire-to-wire spacing, even when the vias are present.



In the following example, a slightly tighter spacing of .24 µm is needed if the other object is less than or equal to 1.0 µm in length.

```
SPACING 0.28 ;
SPACING 0.24 LENGTHTHRESHOLD 1.0 ;
```

The `USELENGTHTHRESHOLD` argument specifies that the threshold spacing rule should be applied if the other object meets the previous `LENGTHTHRESHOLD` value.

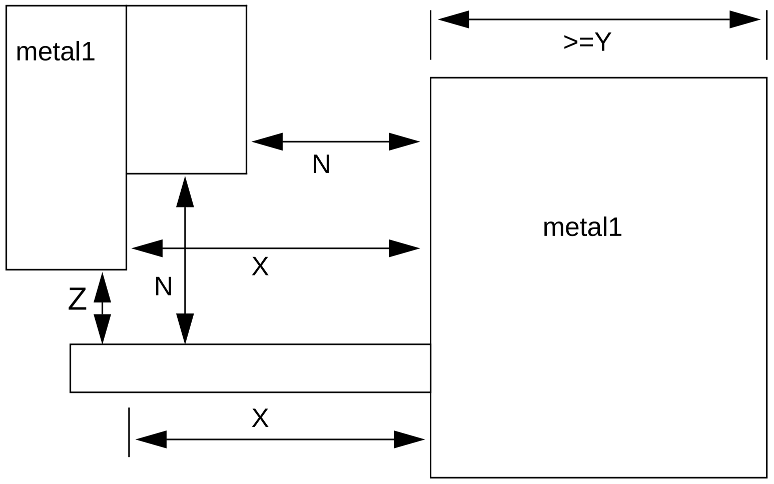
In the following example, a larger spacing of 0.32 µm is needed for wire widths between 1.5 and 9.99 µm. However, if the other object is less than or equal to 1.0 µm in length, the smaller .0.28 µm spacing is applied.

```
SPACING 0.28 ;                    #Default minimum spacing is >=0.28 um.
SPACING 0.28 LENGTHTHRESHOLD 1.0 ;  #For short parallel lengths of <= 1.0 um,
                                    #0.28 spacing is allowed.
SPACING 0.32 RANGE 1.05 9.99 USELENGTHTHRESHOLD ;
                                    #Wide wires with 1.5 <= width <=9.99 need
                                    #0.32 spacing unless the parallel run
                                    #length is <= 1.0 from the previous rule.
```

Influence spacing rules are used to support the inheritance of wide wire spacing by nets connected to the wide wires. For example, a larger spacing is needed for stub wires attached to large objects like pre-routed power wires. A piece of metal connecting to a wider wire will inherit spacing rules for a user-defined distance from the wider wire.

In Figure 1-8 on page 59, a minimum space of N is required between two metal lines when at least one metal line has a width that is >= Y. This spacing must be maintained for any small piece of metal (<Y) that is connected to the wide metal within X range of the wide metal. Outside of this range, normal spacing rules (Z) apply.

Figure 1-8



In the following example, the 0.5 µm spacing applies for the first 2.0 µm of the stub sticking out from the large object. This rule only applies to the stub wire; the previous rule must be included for the wide wire spacing.

```
SPACING 0.28 ;              #Minimum spacing is >= 0.28 um.
SPACING 0.5 RANGE 2.01 2000.0 INFLUENCE 1.000 ;
                            #Stub wires <= 1.0 um from wide wires >= 2.01
                            #require 0.5 um spacing.
```

Some processes only need the `INFLUENCE` rule for certain widths of the stub wire. In the following example, the 0.5 µm spacing is required only for stub wires between 0.5 and 1.0 µm in width.

```
SPACING 0.28 ;         #Minimum spacing is >= 0.28 um.
SPACING 0.5 RANGE 2.01 2000.0 INFLUENCE 1.00 RANGE 0.5 1.0 ;
                       #Stub wires with 0.5 <= width <= 1.0, and <= 1.0 um from
                       #wide wide wires >= 2.01 require 0.5 um spacing.
```

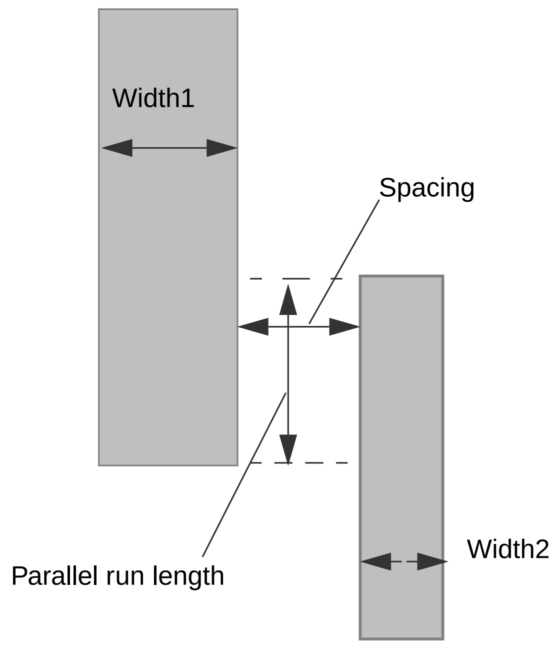
Using Spacing Tables

Some processes have complex width and length threshold rules. Instead of creating multiple `SPACING` rules with different `LENGTHTHRESHOLD` and `RANGE` statements, you can define the information in a spacing table.

For example, for Figure 1-9 on page 60, a typical 90 nm DRC manual might have the following rules described:

Minimum spacing	0.15 µm spacing
Either width>0.25 µm and parallel length>0.50 µm	0.20 µm spacing
Either width>1.50 µm and parallel length>0.50 µm	0.50 µm spacing
Either width>3.00 µm and parallel length>3.00 µm	1.00 µm spacing
Either width>5.00 µm and parallel length>5.00 µm	2.00 µm spacing

Figure 1-9



These rules translate into the following `SPACINGTABLE PARALLELRUNLENGTH` statement:

```
LAYER metal1
    ...
    SPACINGTABLE
     PARALLELRUNLENGTH 0.00 0.50 3.00 5.00        #lengths must be increasing
       WIDTH 0.00      0.15 0.15 0.15 0.15        #max width>0.00
```

```
      WIDTH 0.25      0.15 0.20 0.20 0.20        #max width>0.25
      WIDTH 1.50      0.15 0.50 0.50 0.50        #max width>1.50
      WIDTH 3.00      0.15 0.50 1.00 1.00        #max width>3.00
      WIDTH 5.00      0.15 0.50 1.00 2.00 ;      #max width>5.00
    ...
END metal1
```

Using the `SPACINGTABLE PARALLELRUNLENGTH` statement, the rules can be described in the following way:

1. Find the maximum width of the two objects.
2. Find the table row that matches the maximum width.
3. Find the table column that matches the parallel run length. The value listed where the row and column intersect is the required spacing for that maximum width and parallel run length.

By definition, the width is the smaller dimension of the object (that is, the width of each object must be less than or equal to its length).

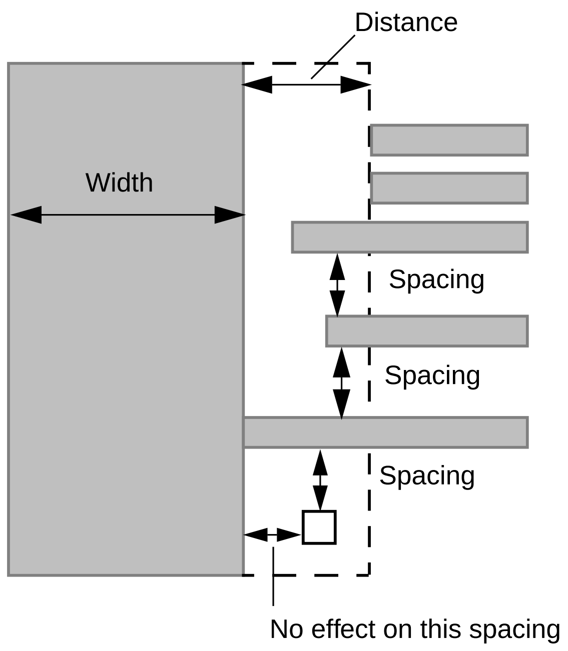
Processes often require a second spacing table to enforce the wide wire spacing rules between nearby perpendicular wires, even if the wires are narrow. Figure 1-10 on page 62 illustrates this situation. Use the following `SPACINGTABLE INFLUENCE` syntax to describe this table:

```
SPACINGTABLE INFLUENCE
     {WIDTH width WITHIN distance SPACING spacing} ... ;
```

If a wire has a width that is greater than *`width`*, and the distance between it and two other wires is less than *`distance`*, the other wires must be separated by spacing that is greater than or equal to *`spacing`*. Typically, the *`distance`* and *`spacing`* values are the same. Note that the distance halo extends horizontally, but not into the corners.

By definition, the width is the smaller dimension of the object (that is, the width is less than or equal to the length of the large wire).

Figure 1-10



The wide wire rules often match the larger width and spacing values in the `SPACINGTABLE PARALLELRUNLENGTH` values. The previously described rules translate into the following `SPACINGTABLE INFLUENCE` statement:

```
LAYER metal1
    ...
    SPACINGTABLE INFLUENCE
      WIDTH 1.50 WITHIN 0.50 SPACING 0.50 #w>1.50, dist<0.50, needs sp>=0.50
      WIDTH 3.00 WITHIN 1.00 SPACING 1.00 #widths must be increasing
      WIDTH 5.00 WITHIN 2.00 SPACING 2.00 ;
    ...
END metal1
```

Macro

```
MACRO macroName
     [CLASS
       { COVER [BUMP]
       | RING
       | BLOCK [BLACKBOX | SOFT]
       | PAD [INPUT | OUTPUT |INOUT | POWER | SPACER | AREAIO]
       | CORE [FEEDTHRU | TIEHIGH | TIELOW | SPACER | ANTENNACELL | WELLTAP]
       | ENDCAP {PRE | POST | TOPLEFT | TOPRIGHT | BOTTOMLEFT | BOTTOMRIGHT}
       }
     ;]
     [FOREIGN foreignCellName [pt [orient]] ;] ...
     [ORIGIN pt ;]
     [EEQ macroName ;]
     [SIZE width BY height ;]
     [SYMMETRY {X | Y | R90} ... ;]
     [SITE siteName [sitePattern] ;] ...
     [PIN statement] ...
     [OBS statement] ...
     [DENSITY statement] ...
     [PROPERTY propName propVal ;] ...

END macroName
```

Defines macros in the design.

`CLASS`

Specifies the macro type. If you do not specify `CLASS`, the macro is considered a `CORE` macro, and a warning prints when the LEF file is read in. You can specify macros of the following types:

`COVER` — Macro with data that is fixed to the floorplan and cannot change, such as power routing (ring pins) around the core. The placers understand that `CLASS COVER` cells have no active devices (such as diffusion or polysilicon), so the `MACRO SIZE` statement does not affect the placers, and you do not need an artificial `OVERLAP` layer. However, any pin or obstruction geometry in the `COVER` cells can affect the pin access checks done by the placers.

A cover macro can be of the following sub-class:

`BUMP`—A physical-only cell that has bump geometries and pins. Typically a bump cell has geometries only on the top-most “bump” metal layer, although it might contain a via and pin to the metal layer below. Bump cells cannot be moved; however, you can change the assignment of a bump cell pin to a driver cell.

`RING` — Large macro that can cut prerouted special nets and connect to these nets with ring pins.

`BLOCK`	Predefined macro used in hierarchical design. A block macro can have one of the following sub-classes: `BLACKBOX`—A block that sometimes only contains a `SIZE` statement that estimates its total area. A blackbox can optionally contain pins, but in many cases, the pin names are taken from a Verilog description and do not need to match the LEF `MACRO` pin names. `SOFT`—A cell that also contains a version of the sub-block that is not fully implemented. Normally, a soft block LEF can still have certain parts of it modified (for example, the aspect ratio, or pin locations) because the sub-block is not yet fully implemented. Any changes should be passed to the sub-block implementation. In contrast, a `BLACKBOX` has no sub-block implementation available.
`PAD`	I/O pad. A pad can be one of the following types: `INPUT`, `OUTPUT`, `INOUT`, `POWER`, or `SPACER`, for I/O rows; `INPUT`, `OUTPUT`, `INOUT`, or `POWER`, for I/O corner pads; `AREAIO` for area I/O driver cells that are placed inside the core of the chip. For an example of a macro pad cell, see Example 1-14 on page 65.

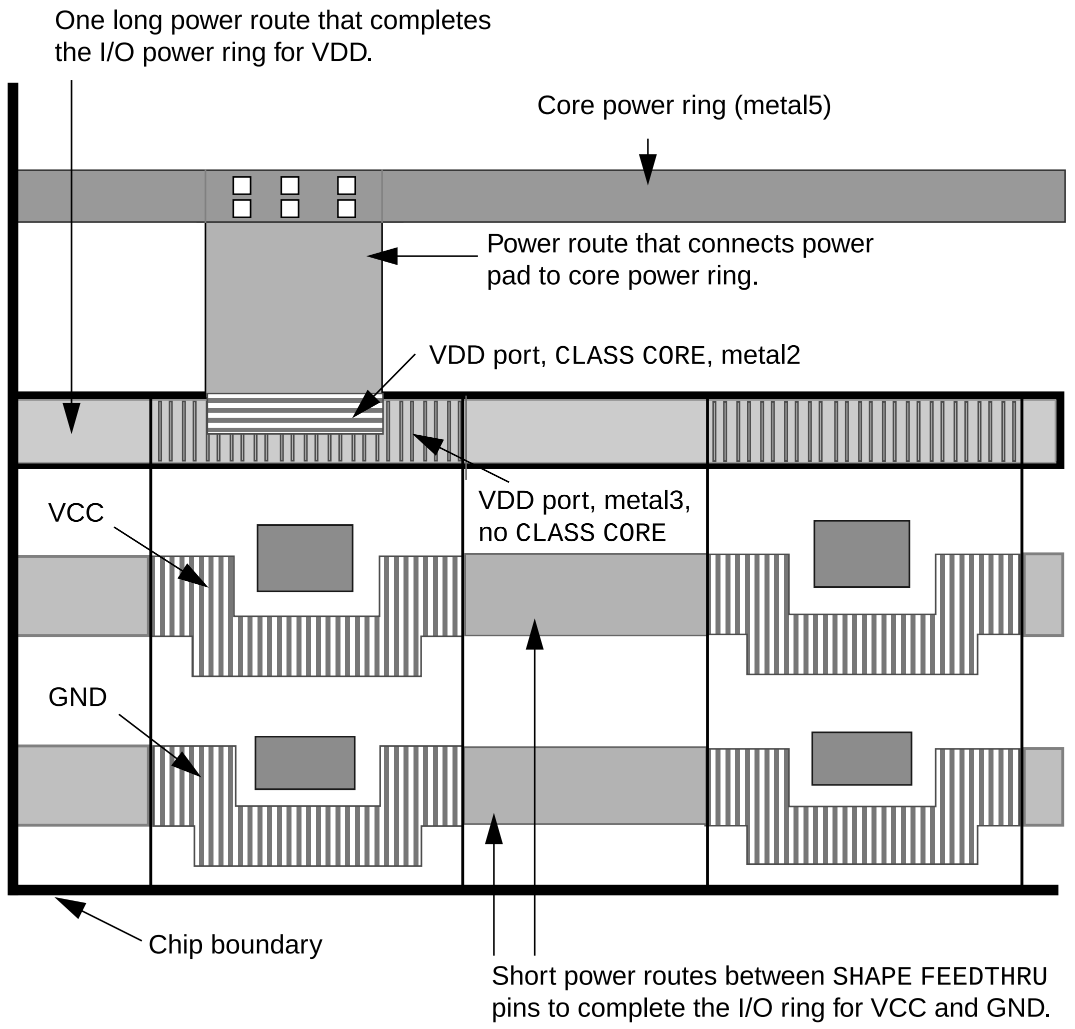
`CORE`	Macro used in the core area. A core macro can be one of the following types: `FEEDTHRU`—Used for connecting to another cell. `TIEHIGH`,`TIELOW`—Used for connecting unused I/O terminals to the power or ground bus. `SPACER`—Sometimes called a filler cell, this cell is used to fill in space between regular core cells. `ANTENNACELL`—Used for solving process antenna violations. This cell has a single input to a diode to bleed off charge that builds up during manufacturing. `WELLTAP`—Standard cell that connects N and P diffusion wells to the correct power or ground wire.
`ENDCAP`	A macro placed at the ends of core rows (to connect with power wiring). If the library includes only one corner I/O macro, then appropriate `SYMMETRY` must be included in its macro description. An `ENDCAP` macro can be one of the following types: `PRE`—A left-end macro `POST`—A right-end macro `TOPLEFT`—A top left I/O corner cell `TOPRIGHT`—A top right I/O corner cell `BOTTOMLEFT` —A bottom left I/O corner cell `BOTTOMRIGHT`—A bottom right I/O corner cell

Example 1-14 Macro Pad Cell

The following example defines a power pad cell that illustrates when to use the CLASS CORE keywords on power ports. For the VDD pin, there are two ports: one to connect to the interior core power ring, and one to complete the I/O power ring. Figure 1-1 on page 6 illustrates this pad cell.

```
MACRO PAD_0
    CLASS PAD ;
    FOREIGN PAD_0 0.000 0.000 ;
    ORIGIN 0.000 0.000 ;
    SIZE 100.000 BY 300.000 ;
    SYMMETRY X Y R90 ;
    SITE PAD_SITE ;

# Define pin VDD with SHAPE ABUTMENT because there are no obstructions
# to block a straight connection to the pad rings. The port without
# CLASS CORE is used for completing the I/O power ring.

    PIN VDD
        DIRECTION INOUT ;
        USE POWER ;
        SHAPE ABUTMENT ;
        PORT
           LAYER metal2 ;
             RECT 0.000 250.000 100.000 260.000 ;
           LAYER metal3 ;
             RECT 0.000 250.000 100.000 260.000 ;
    END

# Define VDD port with PORT CLASS CORE to indicate that the port connects
# to the core area instead of to the pad ring.

      PORT
           CLASS CORE ;
           LAYER metal2 ;
             RECT 0.000 290.000 100.000 300.000 ;
           LAYER metal3 ;
             RECT 0.000 290.000 100.000 300.000 ;
      END
    END VDD

# Define pins VCC and GND with SHAPE FEEDTHRU because these pins
# cannot make a straight connection to the pad rings due to obstructions.
    PIN VCC
          DIRECTION INOUT ;
          USE POWER ;
          SHAPE FEEDTHRU ;
          PORT
            LAYER metal2 ;
              RECT 0.000 150.000 20.000 160.000 ;
              RECT 20.000 145.000 80.000 155.000 ;
              RECT 80.000 150.000 100.000 160.000 ;
            LAYER metal3 ;
              RECT 0.000 150.000 20.000 160.000 ;
              RECT 20.000 145.000 80.000 155.000 ;
              RECT 80.000 150.000 100.000 160.000 ;
          END
    END VCC
    PIN GND
        DIRECTION INOUT ;
        USE GROUND ;
        SHAPE FEEDTHRU ;
        PORT
           LAYER metal2 ;
             RECT 0.000 50.000 20.000 60.000 ;
             RECT 80.000 50.000 100.000 60.000 ;
        END
    END GND
    OBS
        LAYER metal1 ;
            RECT 0.000 0.000 100.000 300.000 ;
        LAYER metal2 ;
            RECT 25.000 50.000 75.000 60.000 ;
            RECT 30.500 157.000 70.500167.000 ;
    END
END PAD_0
```

Figure 1-11 Power Pad Cell


One long power route that completes the I/O power ring for VDD.
Core power ring (metal5)
Power route that connects power pad to core power ring.
VDD port, CLASS CORE, metal2
VDD port, metal3, no CLASS CORE
VCC
GND
Chip boundary
Short power routes between SHAPE FEEDTHRU pins to complete the I/O ring for VCC and GND.

`DENSITY` *`statement`*

Specifies the metal density for large macros.

The `DENSITY` rectangles on a layer should not overlap, and should cover the entire area of the macro. You can choose the size of the rectangles based on the uniformity of the density of the block. If the density is uniform, a single rectangle can be used. If the density is not very uniform, the size of the rectangles can be specified to be 10 to 20 percent of the density window size, so that any error due to non-uniform density inside each rectangle area is small.

For example, if the metal density rule is for a 100 µm x 100 µm window, the density rectangles can be 10x10 µm squares. Any non-uniformity will have little impact on the density calculation accuracy.

If two adjacent rectangles have the same or similar density, they can be merged into one larger rectangle, with one average density value. The choice between accuracy and abstraction is left to the abstract generator.

The `DENSITY` syntax is defined as follows:

```
[DENSITY
   {LAYER layerName ;
      {RECT x1 y1 x2 y2 densityValue ;} ...
   } ...
END] ...
```

densityValue	Specifies the density for the rectangle, as a percentage. For example, `50.0` indicates that the rectangle has a density of 50 percent on *layerName*. *Type:* Float *Value:* `0` to `100`
layerName	Specifies the layer on which to create the rectangle.
x1 y1 x2 y2	Specifies the coordinates of a rectangle. *Type:* Float, specified in microns

Example 1-15 Macro Density

The following statement specifies the density for macro `testMacro`:

```
MACRO testMacro
    CLASS ...
    PIN ...
    OBS ...
    DENSITY
        LAYER metal1 ;
        RECT 0 0 100 100 45.5 ; #rect from (0,0) to (100,100), density of 45.5%
        RECT 100 0 200 100 42.2 ; #rect from (100,0) to (200, 100), density of 42.2%
    END
    ...
```

```
END testMacro
```

EEQ *macroName*

Specifies that the macro being defined should be electrically equivalent to the previously defined *macroName*. EEQ macros include devices such as OR-gates or inverters that have several implementations with different shapes, geometries, and orientations.

Electrically equivalent macros have the following requirements:

- Corresponding pins must have corresponding functionality.
- Pins must be defined in the same order.
- For each group of corresponding pins (one from each macro), pin function and electrical characteristics must be the same.
- The EEQ *macroName* specified must refer to a previously defined macro. If the EEQ *macroName* referenced is already electrically equivalent to other model macros, all referenced macros are considered electrically equivalent.

FOREIGN *foreignCellName* [*pt* [*orient*]]

Specifies the foreign (GDSII) structure name to use when placing an instance of the macro. The optional *pt* coordinate specifies the macro origin (lower left corner when the macro is in north orientation) offset from the foreign origin. The `FOREIGN` statement has a default offset value of `0 0`, if *pt* is not specified.

The optional *orient* value specifies the orientation of the foreign cell when the macro is in north orientation. The default *orient* value is N (North).

Example 1-16 Foreign Statements

The following examples show two variations of the `FOREIGN` statement. The negative offset specifies that the GDSII structure should be above and to the right of the macro lower left corner.

```
    MACRO ABC ...
    FOREIGN ABC -2 -3 ;
```

The positive offset specifies that the GDSII structure should be below and to the left of the macro lower left corner.

```
    MACRO EFG ...
    FOREIGN EFG 2 3 ;
```

`MACRO` *`macroName`*

Specifies the name of the library macro.

`OBS` *`statement`*

Defines obstructions on the macro. Obstruction geometries are specified using layer geometries syntax. See "Macro Obstruction Statement" on page 80 for syntax information.

`ORIGIN` *`pt`*

Specifies how to find the origin of the macro to align with a DEF `COMPONENT` placement point. If there is no `ORIGIN` statement, the DEF placement point for a North-oriented macro is aligned with 0, 0 in the macro. If `ORIGIN` is given in the macro, the macro is shifted by the `ORIGIN` x, y values first, before aligning with the DEF placement point. For example, if the `ORIGIN` is 0, -1, then macro geometry at 0, 1 are shifted to 0, 0, and then aligned to the DEF placement point.

`PIN` *`statement`*

Defines pins for the macro. See "Macro Pin Statement" on page 83 for syntax information.

`PROPERTY` *`propName propVal`*

Specifies a numerical or string value for a macro property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`SITE` *`siteName`* `[`*`sitePattern`*`]`

Specifies the site associated with the macro. Normal row-based standard cells only have a single `SITE` *`siteName`* statement, without a *`sitePattern`*. The *`sitePattern`* syntax indicates that the cell is a gate-array cell, rather than a row-based standard cell. Gate-array standard cells can have multiple `SITE` statements, each with a *`sitePattern`*.

The *`sitePattern`* syntax is defined as follows:

[*xOrigin yOrigin siteOrient* [*stepPattern*]]

xOrigin yOrigin	Specifies the origin of the site inside the macro. *Type:* Float, specified in microns
siteOrient	Specifies the orientation of the site at that location. *Value:* N, S, E, W, FN, FS, FE, or FW

Note: Legal placement locations for macros with site patterns must match the site pattern inside the macro to the site pattern in the design rows.

If the site is repeated, you can specify a *stepPattern* that defines the repeating pattern. The *stepPattern* syntax is defined as follows:

[DO *xCount* BY *yCount* STEP *xStep yStep*]

xCount yCount	Specifies the number of sites to add in the x and y directions. You must specify values that are greater than or equal to 0 (zero). *Type:* Integer
xStep yStep	Specifies the spacing between sites in the x and y directions. *Type:* Float, specified in microns

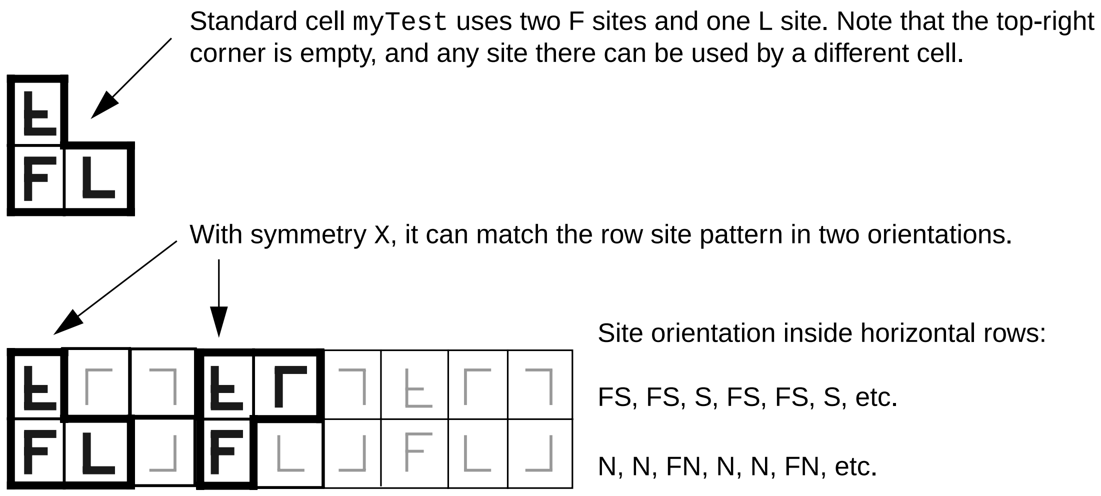
Example 1-17 Macro Site

The following statement defines a macro that uses the sites created in Example 1-29 on page 102:

```
MACRO myTest
    CLASS CORE ;
    SIZE 10.0 BY 14.0 ;   #Uses 2 F and 1 L site, is F + L wide, and double height
    SYMMETRY X ;          #Can flip about the X axis
    SITE Fsite 0 0 N ;    #The lower left Fsite at 0,0
    SITE Fsite 0 7.0 FS ; #The flipped south Fsite above the first Fsite at 0,7
    SITE Lsite 4.0 0 N ;  #The Lsite to the right of the first Fsite at 4,0
    ...
    PIN ... ;
END myTest
```

Figure 1-12 on page 73 illustrates the placement results of this definition.

Figure 1-12


Standard cell myTest uses two F sites and one L site. Note that the top-right corner is empty, and any site there can be used by a different cell.
With symmetry X, it can match the row site pattern in two orientations.
Site orientation inside horizontal rows:
FS, FS, S, FS, FS, S, etc.
N, N, FN, N, N, FN, etc.

The following statement includes the gate-array site pattern syntax. It uses two F sites in a row with N (North) orientation.

```
MACRO myTest
    CLASS CORE ;
    SIZE 8.0 BY 7.0 ; #Width = 2 * Fsite width, height = Fsite height
    SITE Fsite 0 0 N DO 2 BY 1 STEP 4.0 0 ; #Xstep = 4.0 = Fsite width
    ...
END myTest
```

This definition produces a cell with the sites shown in Figure 1-13 on page 73.

Figure 1-13

`SIZE` *`width`* `BY` *`height`* Specifies a placement bounding rectangle, in microns, for the macro. The bounding rectangle always stretches from (0, 0) to the point defined by `SIZE`. For example, given `SIZE 10 BY 40`, the bounding rectangle reaches from (0, 0) after adjustment due to the `ORIGIN` statement, to (100, 400).

Placers assume the placement bounding rectangle cannot overlap placement bounding rectangles of other macros, unless `OBS OVERLAP` shapes are used to create a non-rectangular area.

After placement, a DEF `COMPONENTS` placement *`pt`* indicates where the lower-left corner of the placement bounding rectangle is placed after any possible rotations or flips. The bounding rectangle width and height should be a multiple of the placement grid to allow for abutting cells.

For blocks, the placement bounding rectangle typically contains all pin and blockage geometries, but this is not required. For example, typical standard cells have pins that lie outside the bounding rectangle, such as power pins that are shared with cells in the next row above them.

`SYMMETRY {X | Y | R90}`

Specifies which macro orientations should be attempted by the placer before matching to the site of the underlying rows. In general, most standard cell macros should have symmetry `X` Y. N (North) is always a legal candidate. For each type of symmetry defined, additional orientations become legal candidates. For more information on defining symmetry, see “Defining Symmetry” on page 75.

Possible orientations include:

`X`	N and FS orientations should be tried.
`Y`	N and FN orientations should be tried.
`X Y`	N, FN, FS, and S orientations should all be tried.
`R90`	Specify this value only for pad cells. Do not specify this value for standard cells.

Note: If you do not specify a `SYMMETRY` statement, only N orientation is tried.

For corner I/O pads, if the library includes `BOTTOMLEFT`, `BOTTOMRIGHT`, `TOPLEFT`, and `TOPRIGHT` I/O corner cells, then they are placed in North orientation (no flipping). However, if the library includes only one type of corner I/O, then `SYMMETRY` in x and y are required to create the rows for all four of them.

Defining Cover Macros

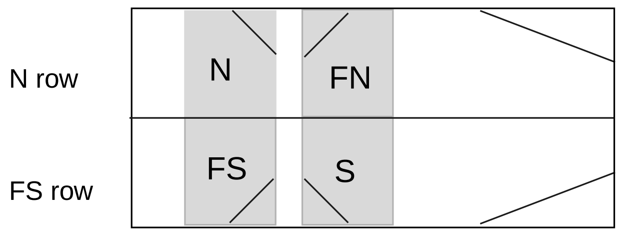
If you define a cover macro with its actual size, some place-and-route tools cannot place the rest of the cells in your design because it uses the cell boundary to check for overlaps. You can resolve this in two ways:

- The easiest way to support a cover macro is to define the cover macro with a small size, for example, 1 by 1.
- If you want to define the cover macro with its actual size, create an overlap layer with the nonrouting `LAYER TYPE OVERLAP` statement. You define this overlap layer (cover macro) with the macro obstruction (`OBS`) statement.

Defining Symmetry

Symmetry statements specify legal orientations for sites and macros. Figure 1-14 on page 75 illustrates the normal orientations for single-height, flipped and abutted rows with standard cells and sites.

Figure 1-14 Normal Orientations for Single-Height Rows

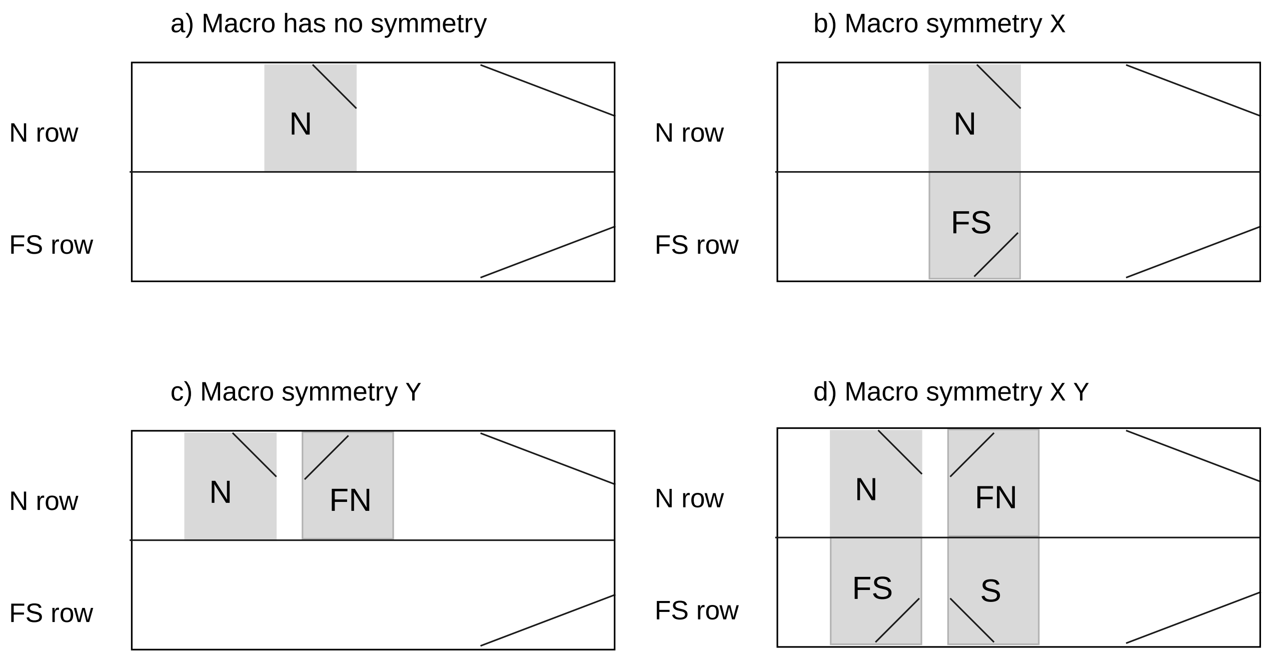


The following examples describe typical combinations of orientations for standard cells. Applications typically create only N (or FS for flipped) row orientations for horizontal standard cell rows; therefore, the examples describe these two rows.

Example 1-18 Single-Height Cells

Single-height cells for flipped and abutted rows should have `SITE` symmetry `Y` and `MACRO` symmetry `X Y`. These specifications allow N and FN macros in N rows, and FS and S macros in FS rows (see d in Figure 1-15 on page 76). These symmetries work with flipped and abutted rows, as well as rows that are not flipped and abutted. If the rows are not flipped, the cells all have N orientation. The extra `MACRO` symmetry of `X` is not required in this case, but causes no problems.

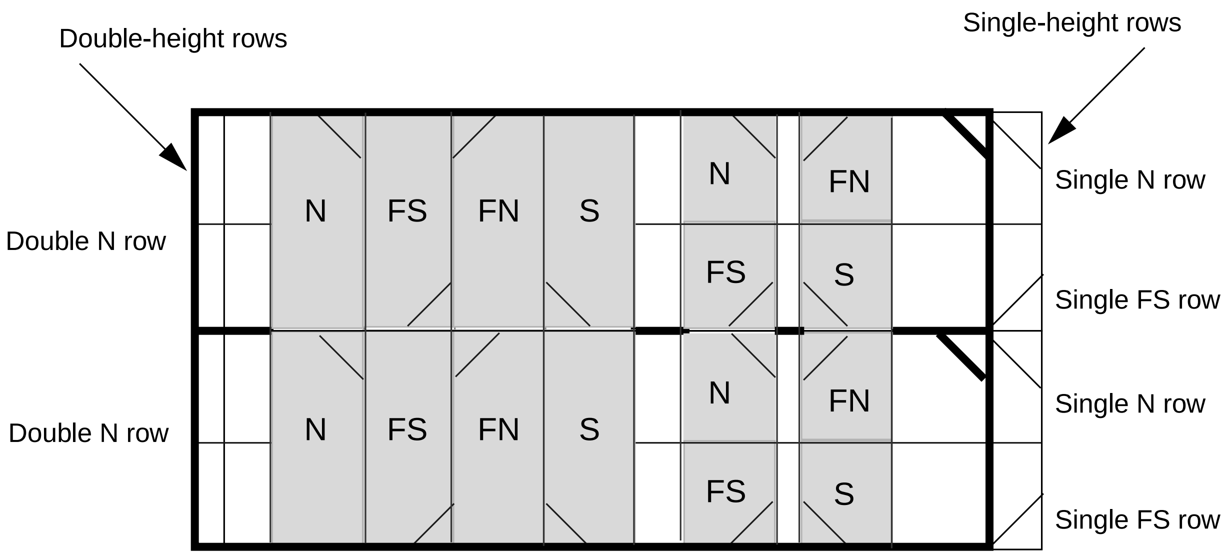
Figure 1-15 Legal Placements for Row Sites with Symmetry Y



Example 1-19 Double-Height Cells

Double-height cells that are intended to align with flipped and abutted single-height rows should have `SITE` symmetry `X Y` and `MACRO` symmetry `X Y`. These symmetries allow all four cell orientations (N, FN, FS, and S) to fit inside the double-height row (see Figure 1-16 on page 77). Usually, double-height rows are just N orientation rows that are abutted and aligned with a pair of single-height flipped and abutted rows.

Figure 1-16 Legal Placements for Single-Height Row Sites with Symmetry Y and Double-Height Row Sites with Symmetry X Y



Note: The single-height rows are shifted slightly to the right of the double-height rows in the above figure for illustration purposes. In a real design, they should be aligned.

Example 1-20 Special Orientations

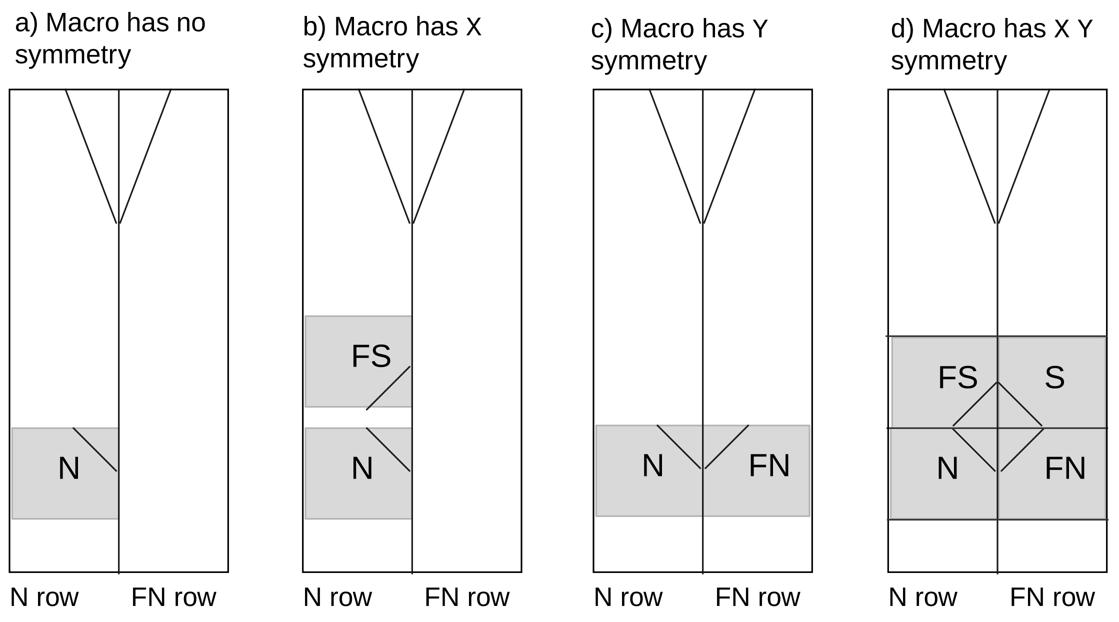
Some single-height cells have special orientation needs. For example, the design requires flipped and abutted rows, but only N and FS orientations are allowed because of the special layout of well taps on the right side of a group of cells that borrow from the left side of the next cell. That is, you cannot place an N and FN cell against each other in one row because only N cells are allowed in an N row. In this case, the `SITE` symmetry should not be defined, and the `MACRO` symmetry should be X. A `MACRO` symmetry of X Y can also be defined because the Y-flipped macros (FN and S orientations) do not match the N or FS rows. This `MACRO` symmetry also works if there are no flipped rows, and only N rows.

Example 1-21 Vertical Rows

Vertical rows use N or FN row and site orientations. The flipped, abutted vertical row orientation is N and FN, rather than the horizontal row orientation of N and FS. Otherwise, the meaning of the site symmetries and macro symmetries is the same as those for horizontal rows.

Single-height sites are normally given symmetry X, and single-height cells are normally given symmetry X Y. Example d in Figure 1-17 on page 78 shows the legal placement for a site with symmetry X, and the typical standard cell `MACRO` symmetry X Y.

Figure 1-17 Legal Placements for Vertical Row Sites With Symmetry X



Layer Geometries

```
{ LAYER layerName [SPACING minSpacing | DESIGNRULEWIDTH value] ;
    [WIDTH width ;]
    { PATH pt ... ;
    | PATH ITERATE pt ... stepPattern ;
    | RECT pt pt ;
    | RECT ITERATE pt pt stepPattern ;
    | POLYGON pt pt pt pt ... ;
    | POLYGON ITERATE pt pt pt pt ... stepPattern ;
    } ...
| VIA pt viaName ;
| VIA ITERATE pt viaName stepPattern ;
} ...
```

Used in the macro obstruction (`OBS`) and pin port (`PIN`) statements to define layer geometries in the design.

`DESIGNRULEWIDTH` *`value`*

Specifies the effective design rule width. If specified, the obstruction or pin is treated as a shape of this width for all spacing checks. If you specify `DESIGNRULEWIDTH`, you cannot specify the `SPACING` argument.
Type: Float

`ITERATE`	Creates an array of the `PATH`, `RECT`, `POLYGON`, or `VIA` geometry, as specified by the given step pattern. `ITERATE` specifications simplify the definitions of cover macros. The syntax for *`stepPattern`* is defined as follows: `DO` *`numX`* `BY` *`numY`* `STEP` *`spaceX spaceY`* *`numX`* — Specifies the number of columns of points. *`numY`* — Specifies the number of rows of points. *`spaceX spaceY`* — Specifies the spacing, in distance units, between the columns and rows of points.
`LAYER` *`layerName`*	Specifies the layer on which to place the geometry. **Note:** For macro obstructions, you can specify cut, implant, or overlap layers.
`PATH` *`pt`*	Creates a path between the specified points, such as *`pt1 pt2 pt3`*. The path automatically extends the length by half of the current *`width`* on both endpoints to form a rectangle. You can also specify a path with a single coordinate, in which case a square whose side is equal to the current *`width`* is placed with its center at *`pt`*.
`POLYGON` *`pt pt pt pt`*	Specifies a sequence of at least four points to generate a polygon geometry. The polygon must be orthogonal (all edges parallel to the x or y axis), so each successive point must vary only one coordinate. Each `POLYGON` statement defines a polygon generated by connecting each successive point, and then by connecting the first and last points.
`RECT` *`pt pt`*	Specifies a rectangle, where the two points specified are opposite corners of the rectangle. There is no functional difference between a geometry specified using `PATH` and a geometry specified using `RECT`.
`SPACING` *`minSpacing`*	Specifies the minimum spacing allowed between this particular obstruction or pin and any other shape. This value overrides the normal `LAYER`-based spacing rules, including wide wire spacing rules. Therefore, a `SPACING` value of 0.1 µm specifies that all other shapes must be spaced at least 0.1 µm away, including

	large width objects that might normally require even more spacing. If you specify `SPACING`, you cannot specify the `DESIGNRULEWIDTH` argument.
`VIA` *`pt viaName`*	Specifies the via to place, and the placement location.
`WIDTH` *`width`*	Specifies the width that the `PATH` statements use. If you do not specify *`width`*, the default width for that layer is used. When you specify a width, that width remains in effect until the next `WIDTH` or `LAYER` statement. When another `LAYER` statement is given, the `WIDTH` is automatically reset to the default width for that layer.

Example 1-22 Layer Geometries

The following example shows how to define a set of geometries, first by using `ITERATE` statements, then by using individual `PATH`, `VIA` and `RECT` statements.

The following two sets of statements are equivalent:

```
PATH ITERATE 532.0 534 1999.2 534
    DO 1 BY 2 STEP 0 1446 ;
VIA ITERATE 470.4 475 VIABIGPOWER12
    DO 2 BY 2 STEP 1590.4 1565 ;
RECT ITERATE 24.1 1.5 43.5 16.5
    DO 2 BY 1 STEP 20.0 0 ;


PATH 532.0 534 1999.2 534 ;
PATH 532.0 1980 1999.2 1980 ;
VIA 470.4 475 VIABIGPOWER12 ;
VIA 2060.8 475 VIABIGPOWER12;
VIA 470.4 2040 VIABIGPOWER12;
VIA 2060.8 2040 VIABIGPOWER12;
RECT 24.1 1.5 43.5 16.5 ;
RECT 44.1 1.5 63.5 16.5 ;
```

Macro Obstruction Statement

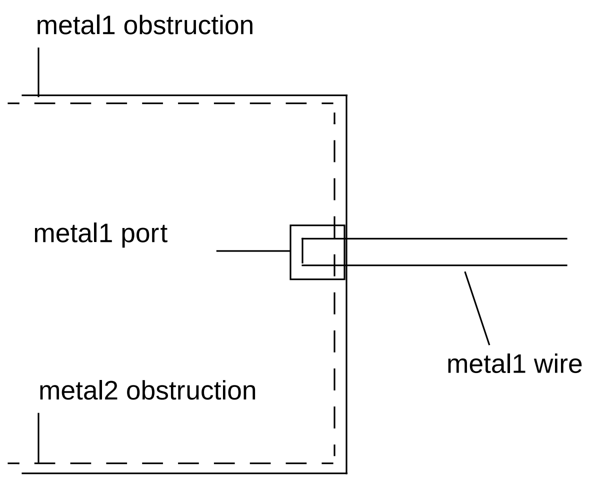
```
[OBS
     { LAYER layerName [SPACING minSpacing | DESIGNRULEWIDTH value] ;
         [WIDTH width ;]
         { PATH pt ... ;
         | PATH ITERATE pt ... stepPattern ;
         | RECT pt pt ;
         | RECT ITERATE pt pt stepPattern ;
         | POLYGON pt pt pt pt ... ;
         | POLYGON ITERATE pt pt pt pt ... stepPattern ;
         } ...
     | VIA pt viaName ;
     | VIA ITERATE pt viaName stepPattern ;
     } ...
END]
```

Defines a set of obstructions (also called blockages) on the macro. You specify obstruction geometries using the layer geometry syntax. See “Layer Geometries” on page 78 for syntax information.

Normally, obstructions block routing, except for when a pin port overlaps an obstruction (a port geometry overrules an obstruction). For example, you can define a large rectangle for a *metal1* obstruction and have *metal1* port in the middle of the obstruction. The port can still be accessed by a via, if the via is entirely inside the port.

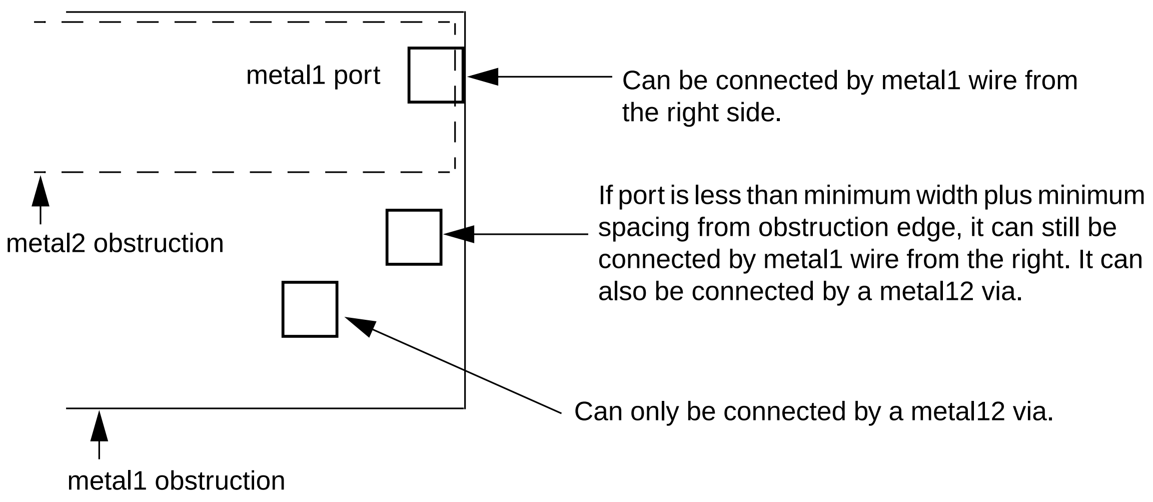
In Figure 1-18 on page 81, the router can only access the *metal1* port from the right. If the *metal2* obstruction did not exist, the router could connect to the port with a *metal12* via, as long as the *metal1* part of the via fit entirely inside the *metal1* port.

Figure 1-18



Routing can also connect to such a port on the same layer if the routing does not cross any obstruction by more than a distance of the total of minimum width plus minimum spacing before reaching the pin. This is because the port geometry is known to be “real,” and any obstruction less than a distance of minimum width plus minimum spacing away from the port is not a real obstruction. If the pin is more than minimum width plus minimum spacing away from the obstruction edge, the router can only route to the pin from the layer above or below using a via (see Figure 1-19 on page 82).

Figure 1-19


metal1 port
Can be connected by metal1 wire from the right side.
If port is less than minimum width plus minimum spacing from obstruction edge, it can still be connected by metal1 wire from the right. It can also be connected by a metal12 via.
metal2 obstruction
Can only be connected by a metal12 via.
metal1 obstruction

If a port is on the edge of the obstruction, a wire can be routed to the port without violations. Pins that are partially covered with obstructions or in apparent violation with nearby obstructions can limit routing options. Even though the violations are not real, the router assumes they are. In these cases, extend each obstruction to cover the pin. The router then accesses the pin as described above.

Benefits of Combining Obstructions

Significant routing time can be saved if obstructions are simplified. Especially in *metal1*, construct obstructions so that free tracks on the layer are accessible to the router. If most of the routing resource is obstructed, simplify the obstruction modeling by combining small obstructions into a single large obstruction. For example, use the bounding box of all *metal1* objects in the cell, rather than many small obstructions, as the bounding box of the obstruction.

You must be sure to model via obstructions over the rest of the cell properly. A single, large *cut12* obstruction over the rest of the cell can do this in some cases, as when *metal1* resource exists within the cell outside the power buses.

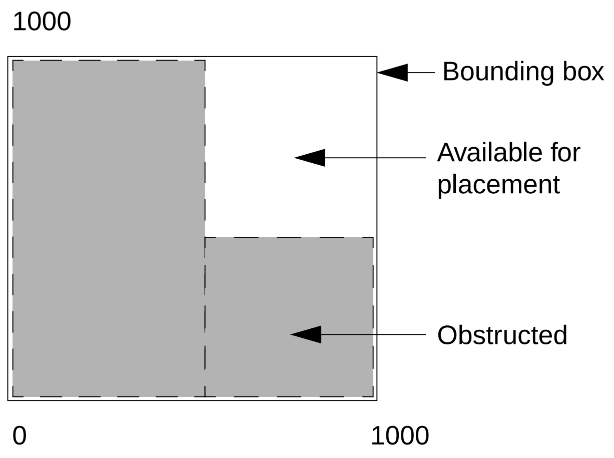
Rectilinear Blocks

Normally, footprint descriptions in LEF are rectangular. However, it is possible to describe rectilinear footprints using an overlap layer. The overlap layer is defined specifically for this purpose and does not contain any routing.

Describe a rectilinear footprint by setting the `SIZE` of the macro as a whole to a rectangular bounding box, then defining obstructions within the bounding box on the overlap layer. The obstructions on the overlap layer indicate areas within the bounding box which no other macro should overlap. The obstructions should completely cover the rectilinear shape of the macro, but not the portion of the bounding box that might overlap with other macros during placement.

Note: Specify the overlaps for the macro using the `OBS` statement. To do this, specify a layer of type `OVERLAP` and then give the overlap geometries, as shown in Figure 1-20 on page 83.

Figure 1-20



```
OBS
     LAYER OVERLAP ;
     RECT 0 0 500 1000 ;
     RECT 500 0 1000 500 ;
END
```

Macro Pin Statement

```
[PIN pinName
     [TAPERRULE ruleName ;]
     [DIRECTION {INPUT | OUTPUT [TRISTATE] | INOUT | FEEDTHRU} ;]
     [USE { SIGNAL | ANALOG | POWER | GROUND | CLOCK } ;]
     [NETEXPR "netExprPropName defaultNetName" ;]
     [SUPPLYSENSITIVITY powerPinName ;]
     [GROUNDSENSITIVITY groundPinName ;]
     [SHAPE {ABUTMENT | RING | FEEDTHRU} ;]
     [MUSTJOIN pinName ;]
     {PORT
       [CLASS {NONE | CORE} ;]
```

```
      {layerGeometries} ...
    END} ...
    [PROPERTY propName propVal ;] ...
    [ANTENNAPARTIALMETALAREA value [LAYER layerName] ;] ...
    [ANTENNAPARTIALMETALSIDEAREA value [LAYER layerName] ;] ...
    [ANTENNAPARTIALCUTAREA value [LAYER layerName] ;] ...
    [ANTENNADIFFAREA value [LAYER layerName] ;] ...
    [ANTENNAMODEL {OXIDE1 | OXIDE2 | OXIDE3 | OXIDE4} ;] ...
    [ANTENNAGATEAREA value [LAYER layerName] ;] ...
    [ANTENNAMAXAREACAR value LAYER layerName ;] ...
    [ANTENNAMAXSIDEAREACAR value LAYER layerName ;] ...
    [ANTENNAMAXCUTCAR value LAYER layerName ;] ...
END pinName]
```

Defines pins for the macro. `PIN` statements must be included in the LEF specification for each macro. All pins, including VDD and VSS, must be specified. The first pin listed becomes the first pin in the database. List the pins in the following order:

- Netlist pins, including inout pins, output pins, and input pins
- Power and ground pins
- Mustjoin pins

`ANTENNADIFFAREA` *`value`* `[LAYER` *`layerName`*`]`

Specifies the diffusion (diode) area, in micron-squared units, to which the pin is connected on a layer. If you do not specify a layer name, the value applies to all layers. For more information on process antenna calculation, see Appendix C, “Calculating and Fixing Process Antenna Violations,”

`ANTENNAGATEAREA` *`value`* `[LAYER` *`layerName`*`]`

Specifies the gate area, in micron-squared units, to which the pin is connected on a layer. If you do not specify a layer name, the value applies to all layers. For more information on process antenna calculation, see Appendix C, “Calculating and Fixing Process Antenna Violations,”

`ANTENNAMAXAREACAR` *`value`* `LAYER` *`layerName`*

For hierarchical process antenna effect calculation, specifies the maximum cumulative area ratio value on the specified *`layerName`*, using the metal area at or below the current pin layer, excluding the pin area itself. This is used to calculate the actual cumulative antenna ratio on the pin layer, or the layer above it.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAMAXCUTCAR` *`value`* `LAYER` *`layerName`*

For hierarchical process antenna effect calculation, specifies the maximum cumulative antenna ratio value on the specified *`layerName`*, using the cut area at or below the current pin layer, excluding the pin area itself. This is used to calculate the actual cumulative antenna ratio for the cuts above the pin layer.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAMAXSIDEAREACAR` *`value`* `LAYER` *`layerName`*

For hierarchical process antenna effect calculation, specifies the maximum cumulative antenna ratio value on the specified *`layerName`*, using the metal side wall area at or below the current pin layer, excluding the pin area itself. This is used to calculate the actual cumulative antenna ratio on the pin layer or the layer above it.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAMODEL {OXIDE1 | OXIDE2 | OXIDE3 | OXIDE4}`

Specifies the oxide model for the pin. If you specify an `ANTENNAMODEL` statement, the value affects all `ANTENNAGATEAREA` and `ANTENNA*CAR` statements for the pin that follow it until you specify another `ANTENNAMODEL` statement. The `ANTENNAMODEL` statement does not affect `ANTENNAPARTIAL*AREA` and `ANTENNADIFFAREA` statements because they refer to the total metal, cut, or diffusion area connected to the pin, and do not vary with each oxide model.
Default: `OXIDE1`, for a new `PIN` statement

Because LEF is often used incrementally, if an `ANTENNA` statement occurs twice for the same oxide model, the last value specified is used.

For most standard cells, there is only one value for the ANTENNAPARTIAL*AREA and ANTENNADIFFAREA values per pin; however, for a block with six routing layers, it is possible to have six different ANTENNAPARTIAL*AREA values and six different ANTENNAPINDIFFAREA values per pin. It is also possible to have six different ANTENNAPINGATEAREA and ANTENNAPINMAX*CAR values for each oxide model on each pin.

Example 1-23 Pin Antenna Model

The following example describes oxide model information for pins IN1 and IN2.

```
MACRO GATE1
    PIN IN1
        ANTENNADIFFAREA 1.0 ;             #not affected by ANTENNAMODEL
        ...
        ANTENNAMODELOXIDE OXIDE1 ;        #OXIDE1 not required, but is good
                                          #practice
        ANTENNAGATEAREA 1.0 ;             #OXIDE1 gate area
        ANTENNAMAXAREACAR 50.0 LAYER m1 ; #metal1 CAR value
        ...
        ANTENNAMODEL OXIDE2 ;             #OXIDE2 starts here
        ANTENNAGATEAREA 3.0 ;             #OXIDE2 gate area
        ...
    PIN IN2
        ANTENNADIFFAREA 2.0 ;             #not affected by ANTENNAMODEL
        ANTENNAPARTIALMETALAREA 2.0 LAYER m1 ;
        ...
        #no OXIDE1 specified for this pin
        ANTENNAMODEL OXIDE2 ;
        ANTENNAGATEAREA 1.0 ;
        ...
```

ANTENNAPARTIALCUTAREA *value* [LAYER *layerName*]

Specifies the partial cut area above the current pin layer and inside the macro cell on the layer. For a hierarchical design, only the cut layer above the I/O pin layer is needed for partial antenna ratio calculation. If you do not specify a layer name, the value applies to all layers.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAPARTIALMETALAREA` *`value`* `[LAYER` *`layerName`*`]`

Specifies the partial metal area connected directly to the I/O pin and the inside of the macro cell on the layer. For a hierarchical design, only the same metal layer as the I/O pin, or the layer above it, is needed for partial antenna ratio calculation. If you do not specify a layer name, the value applies to all layers.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

Note: Metal area is calculated by adding the pin's geometric metal area and the `ANTENNAPARTIALMETALAREA` value.

`ANTENNAPARTIALMETALSIDEAREA` *`value`* `[LAYER` *`layerName`*`]`

Specifies the partial metal side wall area connected directly to the I/O pin and the inside of the macro cell on the layer. For a hierarchical design, only the same metal layer as the I/O pin or the layer above is needed for partial antenna ratio calculation. If you do not specify a layer name, the value applies to all layers.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`CLASS {NONE | CORE}` Specifies whether or not the port is a core port. A core port is only used on power and ground I/O pads. The core port indicates which power or ground port to connect to a core ring for the chip (inside the I/O pads).
Default: `NONE`

`DIRECTION {INPUT | OUTPUT [TRISTATE] | INOUT | FEEDTHRU}`

Specifies the pin type. Most current tools do not usually use this keyword. Typically, pin directions are defined by timing library data, and not from LEF.
Default: `INPUT`
Value: Specify one of the following:

`INPUT`	Pin that accepts signals coming into the cell.

`OUTPUT [TRISTATE]`	Pin that drives signals out of the cell. The optional `TRISTATE` argument indicates tristate output pins for ECL designs.
`INOUT`	Pin that can accept signals going either in or out of the cell.
`FEEDTHRU`	Pin that goes completely across the cell.

`GROUNDSENSITIVITY` *`groundPinName`*

Specifies that if this pin is connected to a tie-low connection (such as `1'b0` in Verilog), it should connect to the same net to which *`groundPinName`* is connected.

`groundPinName` must match a pin on this macro that has a `USE GROUND` attribute. The ground pin definition can follow later in this `MACRO` statement; it does not have to be defined before this pin definition. For an example, see Example 1-24 on page 90.

Note: `GROUNDSENSITIVITY` is useful only when there is more than one ground pin in the macro. By default, if there is only one `USE GROUND` pin, then the tie-low connections are already implicitly defined (that is, tie-low connections are connected to the same net as the one ground pin).

`layerGeometries`

Defines port geometries for the pin. You specify port geometries using layer geometries syntax. See "Layer Geometries" on page 78 for syntax information.

`MUSTJOIN` *`pinName`*

Specifies the name of another pin in the cell that must be connected with the pin being defined. `MUSTJOIN` pins provide connectivity that must be made by the router. In the LEF file, each pin referred to must be defined before the referring pin. The remaining `MUSTJOIN` pins in the set do not need to be defined contiguously.

Note: `MUSTJOIN` pin names are never written to the DEF file; they are only used by routers to add extra connection points during routing.

`MUSTJOIN` pins have the following restrictions:

- A set of `MUSTJOIN` pins cannot have more than one schematic pin.
- Nonschematic `MUSTJOIN` pins must be defined after all other pins.

Schematic and nonschematic `MUSTJOIN` pins are handled in slightly different ways. For schematic `MUSTJOIN` pins, the pins are added to the pin set for the (unique) net associated with the ring for each component instance of the macro. The net is routed in the usual manner, and routing data for the `MUSTJOIN` pins are included in routing data for the net.

The mustjoin routing is not necessarily performed before the rest of the net. Timing relations should not be given for `MUSTJOIN` pins, and internal mustjoin routing is modeled as lumped capacitance at the schematic pin.

Nonschematic `MUSTJOIN` pin sets get routed in the usual manner. However, when the DEF file is outputted, routing data is reported in the `NETS` section of the file as follows:

```
MUSTJOIN compName pinName + regularWiring ;
```

Here, *`compName`* is the component and *`pinName`* is an arbitrary pin in the set. You can also use the preceding to input prewiring for the `MUSTJOIN` pin, using `FIXED` or `COVER`.

`NETEXPR "`*`netExprPropName defaultNetName`*`"`

Specifies a net expression property name (such as `power1` or `power2`) and a default net name. If *`netExprPropName`* matches a net expression property in the netlist (such as in Verilog, VHDL, or OpenAccess), then the property is evaluated, and the software identifies a net to which to connect this pin. If this property does not exist, *`defaultNetName`* is used for the net name.

`netExprPropName` must be a simple identifier in order to be compatible with other languages, such as Verilog and CDL.

Therefore, it can only contain alphanumeric characters, and the first character cannot be a number. For example, `power2` is a legal name, but `2power` is not. You cannot use characters such as `$` and `!`. The *`defaultName`* can be any legal DEF net name.

Example 1-24 Net Expression and Supply Sensitivity

The following statement defines sensitivity and net expression values for four pins on the macro `myMac`:

```
MACRO myMac
    ...
    PIN in1
        ...
        SUPPLYSENSITIVITY vddpin1 ; #If in1 is 1’b1, use net connected to vddpin1.
                                    #Note that no GROUNDSENSITIVITY is needed
                                    #because only one ground pin exists.
                                    #Therefore, 1’b0 implicitly means net from
                                    #pin gndpin.
        ...
    END in1

    PIN vddpin1
        ...
        NETEXPR "power1 VDD1" ; #If power1 net expression is defined in the
                                #netlist, use it to find the net connection. If
                                #not, use net VDD1.
        ...
    END vddpin1
    PIN vddpin2
        ...
        NETEXPR "power2 VDD2" ; #If power2 net expression is defined in the
                                #netlist, use it to find the net connection.If
                                #not, use net VDD2.
        ...
    END vddpin2
    PIN gndpin
        ...
        NETEXPR "gnd1 GND" ; #If gnd1 net expression is defined in the
                             #netlist, use  it to find the net connection. If
                             #not, use net GND.
        ...
    END gndpin
    ...
```

```
END myMac
```

`PIN` *`pinName`*	Specifies the name for the library pin.
`PORT`	Starts a pin port statement, which defines a collection of geometries that are electrically equivalent points. A pin can have multiple ports. Routers connect to any of the member geometries of a port, without affecting the electrical performance of the circuit.

`PROPERTY` *`propName propVal`*

Specifies a numerical or string value for a pin property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`SHAPE`

Specifies a pin with special connection requirements because of its shape.
Value: Specify one of the following:

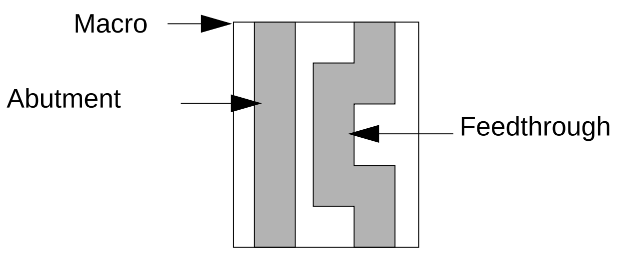
`ABUTMENT`	Pin that goes straight through cells with a regular shape and connects to pins on adjoining cells without routing.
`RING`	Pin on a large block that forms a ring around the block to allow connection to any point on the ring. Cover macro special pins also typically have shape `RING`.
`FEEDTHRU`	Pin with an irregular shape with a jog or neck within the cell.

Figure 1-21 on page 92 shows an example of an abutment and a feedthrough pin.

Note: When you define feedthrough and abutment pins for use with power routing, you must do the following:

- Feedthrough pin widths must be the same on both edges and consistent with the routing width used with the power route commands.
- Feedthrough pin centers on both edges must align for successful routing.
- Power pins in fork shapes must be represented in two ports and be defined as a feedthrough shape. In most other cases, power pin geometries do not represent more than one port.
- An abutment pin must have at least one geometric rectangle with layer and width consistent with the values specified in the power route commands.

Figure 1-21



`SUPPLYSENSITIVITY` *`powerPinName`*

Specifies that if this pin is connected to a tie-high connection (such as `1'b1` in Verilog), it should connect to the same net to which *`powerPinName`* is connected.

`powerPinName` must match a pin on this macro that has a `USE POWER` attribute. The power pin definition can follow later in this `MACRO` statement; it does not have to be defined before this pin definition. For an example, see Example 1-24 on page 90.

Note: `SUPPLYSENSITIVITY` is useful only when there is more than one power pin in the macro. By default, if there is only one `USE POWER` pin, then the tie-high connections are already implicitly defined (that is, tie-high connections are connected to the same net as the one power pin).

`TAPERRULE` *`ruleName`*
Specifies the nondefault rule to use when tapering wires to the pin.

`USE {ANALOG | CLOCK | GROUND | POWER | SIGNAL}`
Specifies how the pin is used. Pin use is required for timing analysis.
Default: `SIGNAL`
Value: Specify one of the following:

`ANALOG`	Pin is used for analog connectivity.
`CLOCK`	Pin is used for clock net connectivity.
`GROUND`	Pin is used for connectivity to the chip-level ground distribution network.
`POWER`	Pin is used for connectivity to the chip-level power distribution network.
`SIGNAL`	Pin is used for regular net connectivity.

Manufacturing Grid

```
[MANUFACTURINGGRID value ;]
```

Defines the manufacturing grid for the design. The manufacturing grid is used for geometry alignment. When specified, shapes and cells are placed in locations that snap to the manufacturing grid.

`value`
Specifies the value for the manufacturing grid, in microns. *`value`* must be a positive number.
Type: Float

Maximum Via Stack

```
[MAXVIASTACK value [RANGE bottomLayer topLayer] ;]
```

Specifies the maximum number of single-cut stacked vias that are allowed on top of each other (that is, in one continuous stack). A via is considered to be in a stack with another via if the cut of the first via overlaps any part of the cut of the second via. The `MAXVIASTACK` statement should follow the `LAYER` statements in the LEF file; however, it is not attached to any particular layer. You can specify only one `MAXVIASTACK` statement in a LEF file.

`RANGE` *`bottomLayer topLayer`*	Specifies a range of layers for which the maximum stacked via rule applies. If you do not specify a range, the value applies for all layers.
`value`	Specifies the maximum allowed number of single-cut stacked vias. *Type:* Integer

Example 1-25 Maximum Via Stack Statement

The following `MAXVIASTACK` statement specifies that only four stacked vias are allowed on top of each other. This rule applies to all layers.

```
LAYER metal9
    ...
END LAYER

MAXVIASTACK 4 ;
```

If you specify the following statement instead, the stacked via limit applies only to layers *metal1* through *metal7*.

```
MAXVIASTACK 4 RANGE m1 m7 ;
```

Nondefault Rule

```
[NONDEFAULTRULE ruleName
     [HARDSPACING ;]
     {LAYER layerName
        WIDTH width ;
        [DIAGWIDTH diagWidth ;]
        [SPACING minSpacing ;]
        [WIREEXTENSION value ;]
     END layerName} ...
     [VIA viaStatement] ...
     [USEVIA viaName ;] ...
     [USEVIARULE viaRuleName ;] ...
     [MINCUTS cutLayerName numCuts ;] ...
     [PROPERTY propName propValue ;] ...

END ruleName]
```

Defines the wiring width, design rule spacing, and via size for regular (signal) nets. You do not need to define cut layers for the nondefault rule.

Some tools have limits on the total number of nondefault rules they can store. This limit can be as low as 30; however most tools that support 90 nanometer rules (that is, LEF 5.5 and 5.6) can handle at least 255.

Note: Use the `VIA` statement to define vias for nondefault wiring.

`DIAGWIDTH` *`diagWidth`* Specifies the diagonal width for *`layerName`*, when 45-degree routing is used.
Default: The minimum width value (`WIDTH` *`minWidth`*)
Type: Float, specified in microns

`HARDSPACING` Specifies that any spacing values that exceed the LEF `LAYER` spacing requirements are “hard” rules instead of “soft” rules. By default, routers treat extra spacing requirements as soft rules that are high cost to violate, but not real spacing violations. However, in certain situations, the extra spacing should be treated as a hard, or real, spacing violation, such as when the route will be modified with a post-process that replaces some of the extra space with metal.

`LAYER` *`layerName`* `...` `END` *`layerName`*
Specifies the layer for the various width and spacing values. This layer must be a routing layer.

`MINCUTS` *`cutLayerName numCuts`*
Specifies the minimum number of cuts allowed for any via using the specified cut layer. Routers should only use vias (generated or predefined fixed vias) that have at least *`numCuts`* cuts in the via.
Type: (*`numCuts`*) Positive integer

`NONDEFAULTRULE` *`ruleName`*
Specifies a name for the new routing rule. The name `DEFAULT` is reserved for the default routing rule used by most nets. The default routing rule is constructed automatically from the LEF `LAYER` statement `WIDTH`, `DIAGWIDTH`, `SPACING`, and `WIREEXTENSION` values, from the LEF `VIA` statement (any vias with the `DEFAULT` keyword), and from the LEF `VIARULE` statement (any via rules with the `DEFAULT` keyword). If you specify `DEFAULT` for *`ruleName`*, the automatic creation is

overridden, and the default routing rule is defined directly from this rule definition.

`PROPERTY` *`propName propValue`*

Specifies a numerical or string value for a nondefault rule property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`SPACING` *`minSpacing`*

Specifies the recommended minimum spacing for *`layerName`* of routes using this `NONDEFAULTRULE` to other geometries. If the spacing is given, it must be at least as large as the foundry minimum spacing rules defined in the `LAYER` definitions. Routers should attempt to meet this recommended spacing rule; however, the spacing rule can be relaxed to the foundry spacing rules along some parts of the wire if the routing is very congested, or if it is difficult to reach a pin.

Adding extra space to a nondefault rule allows a designer to reduce cross-coupling capacitance and noise, but a clean route with no actual foundry spacing violations will still be allowed, unless the `HARDSPACING` statement is specified.
Type: Float, specified in microns

`USEVIA` *`viaName`*

Specifies a previously defined via from the LEF `VIA` statement, or a previously defined `NONDEFAULTRULE` via to use with this routing rule.

`USEVIARULE` *`viaRuleName`*

Specifies a previously defined `VIARULE GENERATE` rule to use with this routing rule. You cannot specify a rule from a `VIARULE` without a `GENERATE` keyword.

`VIA` *`viaStatement`*

Defines a new via. You define nondefault vias using the same syntax as default vias. For syntax information, see "Via" on page 107. All vias, default and nondefault, must have unique via names. If you define more than one via for a rule, the router chooses which via to use.

Note: Defining a new via is no longer recommended, and is likely to become obsolete. Instead, vias should be predefined in a LEF `VIA` statement, then added to the nondefault rule using the `USEVIA` keyword.

`WIDTH` *`width`*	Specifies the required minimum width for *`layerName`*. *Type:* Float, specified in microns
`WIREEXTENSION` *`value`*	Specifies the distance by which wires are extended at vias. Enter `0` (zero) to specify no extension. Values other than `0` must be greater than or equal to half of the routing width for the layer, as defined in the nondefault rule. *Default:* Wires are extended half of the routing width *Type:* Float, specified in microns

Example 1-26 Nondefault Rule Statement

Assume two default via rules were defined:

```
VIARULE via12rule GENERATE DEFAULT
    LAYER metal1 ;
    ...
END via12rule
VIARULE via23rule GENERATE DEFAULT
    LAYER metal2 ;
    ...
END via23rule
```

Assuming the minimum width is 1.0 µm, the following nondefault rule creates a 1.5x minimum width wire using default spacing:

```
NONDEFAULTRULE wide1_5x
    LAYER metal1
        WIDTH 1.5 ; #metal1 has a 1.5 um width
    END metal1
    LAYER metal2
        WIDTH 1.5 ;
    END metal2
    LAYER metal3
        WIDTH 1.5 ;
    END metal3
END wide1_5x
```

Note: If there were no default via rules, then a `VIA`, `USEVIA`, or `USEVIARULE` keyword would be required. Because there are none defined, the default via rules are implicitly inherited for this nondefault rule; therefore, `via12rule` and `via23rule` would be used for this routing rule.

The following nondefault rule creates a 3x minimum width wire using default spacing with at least two-cut vias:

```
NONDEFAULTRULE wide3x
    LAYER metal1
        WIDTH 3.0 ; #metal1 has 3.0 um width
    END metal1
    LAYER metal2
        WIDTH 3.0 ;
    END metal2
    LAYER metal3
        WIDTH 3.0 ;
    END metal3
    #viarule12 and viarule23 are used implicitly
    MINCUTS cut12 2 ; #at least two-cut vias are required for cut12
    MINCUTS cut23 2 ;
END wide3x
```

The following nondefault rule creates an “analog” rule with its own special vias, and with hard extra spacing:

```
NONDEFAULTRULE analog_rule
    HARDSPACING ;     #do not let any other signal close to this one
    LAYER metal1
        WIDTH 1.5 ;   #metal1 has 1.5 um width
        SPACING 3.0 ; #extra spacing of 3.0 um
    END metal1
    LAYER metal2
        WIDTH 1.5
        SPACING 3.0
    END metal2
    LAYER metal3
        WIDTH 1.5
        SPACING 3.0
    END metal3
    #Use predefined "analog vias"
    #The DEFAULT VIARULES will not be inherited.
    USEVIA via12_fixed_analog_via ;
    USEVIA via_23_fixed_analog_via ;
END analog_rule
```

Property Definitions

```
[PROPERTYDEFINITIONS
     [objectType propName propType [RANGE min max]
        [value | "stringValue"]
     ;] ...
END PROPERTYDEFINITIONS]
```

Lists all properties used in the LEF file. You must define properties in the `PROPERTYDEFINITIONS` statement before you can refer to them in other sections of the LEF file.

`objectType`	Specifies the object type being defined. You can define properties for the following object types:
	`LAYER`
	`LIBRARY`
	`MACRO`
	`NONDEFAULTRULE`
	`PIN`
	`VIA`
	`VIARULE`

`propName`	Specifies a unique property name for the object type.

`propType`	Specifies the property type for the object type. You can specify one of the following property types:
	`INTEGER`
	`REAL`
	`STRING`

`RANGE` *`min max`*	Limits real number and integer property values to a specified range. That is, the value must be greater than or equal to *`min`* and less than or equal to *`max`*.
`value` `|` `"`*`stringValue`*`"`	Assigns a numeric value or a name to a `LIBRARY` object type.

Note: Assign values to other properties in the section of the LEF file that describes the object to which the property applies.

Example 1-27 Property Definitions Statement

The following example shows library, via, and macro property definitions.

```
PROPERTYDEFINITIONS
    LIBRARY versionNum INTEGER 12;
    LIBRARY title STRING "Cadence96";
    VIA count INTEGER RANGE 1 100;
    MACRO weight REAL RANGE 1.0 100.0;
    MACRO type STRING;
END PROPERTYDEFINITIONS
```

Same-Net Spacing

```
[SPACING
     [SAMENET layerName layerName minSpace [STACK] ;] ...
END SPACING]
```

Defines the same-net spacing rules. Same-net spacing rules determine minimum spacing between geometries in the same net and are only required if same-net spacing is smaller than different-net spacing, or if vias on different layers have special stacking rules. These specifications are used for design rule checking by the routing and verification tools.

Spacing is the edge-to-edge separation, both orthogonal and diagonal. Same-net spacing rules are less restrictive than different-net spacing rules.

Note: If SAMENET rules are not defined, via stacking is legal by default.

layerName layerName	Specify the names of the layers for which the same-net spacing rule applies. You can specify spacing rules for routing layers and cut layers. For a routing layer, the same-net spacing rule is defined by specifying the same layer name twice.
minSpace	Specifies the minimum spacing.
STACK	Allows stacked vias at a routing layer. The STACK keyword is intended for use between two adjacent cut layers. The use of the STACK keyword on metal layers is considered obsolete, and is ignored.

Example 1-28 Same-Net Spacing Rules

The following example sets the minimum same-net spacing for layer *metal1*:

```
SAMENET M1 M1 1.5 ;
```

This statement is only needed if the same-net spacing is different than the different-net spacing already in the *metal1* LAYER section.

The following example defines spacing for cut layers. The statement sets the same-net spacing to 2.3 microns (also in the xy plane) for pairs of features, where one geometry is on *CUT01* and the other geometry is on *CUT12*, and disables stacked (overlapped) vias.

```
SAMENET CUT01 CUT12 2.3 ;
```

You can allow stacked vias at a routing layer by including the STACK keyword in the same-net specification for that layer. For example, the following specification allows a 1-2 via and a 2-3 via to be stacked at the same point in the design. If they do not exactly align, the same-net spacing rule of 1.5 µm is applied.

```
SAMENET CUT12 CUT23 1.5 STACK ;
```

If you want to allow arbitrary stacking between vias, you can either use the default behavior by not specifying a SAMENET rule, or you can do it explicitly using the following statement:

```
SAMENET via12 via23 0.0 STACK ;
```

Nondefault Rules and Same-Net Spacing

NONDEFAULT rules inherit the default SPACING SAMENET rules. However if the NONDEFAULT rule has its own SAMENET rules, these rules override the default values.

For example, no SAMENET rules at all allows stacking and overlap for default and nondefault routes. However, the following default:

```
SAMENET cut12 cut23 0.1 ;
```

turns off via12 and via23 stacking for all default and nondefault routes, and requires the vias to be separated by 0.1 µm.

A default SPACING section with

```
SAMENET cut12 cut23 0.1 ;
```

followed by a NONDEFAULT rule with

```
SAMENET cut12 cut23 0.1 STACK ;
```

allows an exactly aligned stacked via for the nondefault routes, but does not allow stacking for the nondefault routes.

Site

```
SITE siteName
     CLASS {PAD | CORE} ;
     [SYMMETRY {X | Y | R90} ... ;]
     [ROWPATTERN {existingSiteName siteOrient} ... ;]
     SIZE width BY height ;
END siteName
```

Defines a placement site in the design. A placement site gives the placement grid for a family of macros, such as I/O, core, block, analog, digital, short, tall, and so forth. `SITE` definitions can be used in DEF `ROW` statements.

`CLASS {PAD | CORE}` Specifies whether the site is an I/O pad site or a core site.

`ROWPATTERN {`*`previousSiteName siteOrient`*`}`
Specifies a set of previously defined sites and their orientations that together form *`siteName`*.

`existingSiteName`

Specifies the name of a previously defined site. The height of each previously defined site must be the same as the height specified for *`siteName`*, and the sum of the widths of the previously defined sites must equal the width specified for *`siteName`*.

`siteOrient`

Specifies the orientation for the previously defined site. This value must be one of `N`, `S`, `E`, `W`, `FN`, `FS`, `FE`, and `FW`. For more information on orientations, see "Specifying Orientation" in the DEF `COMPONENT` section.

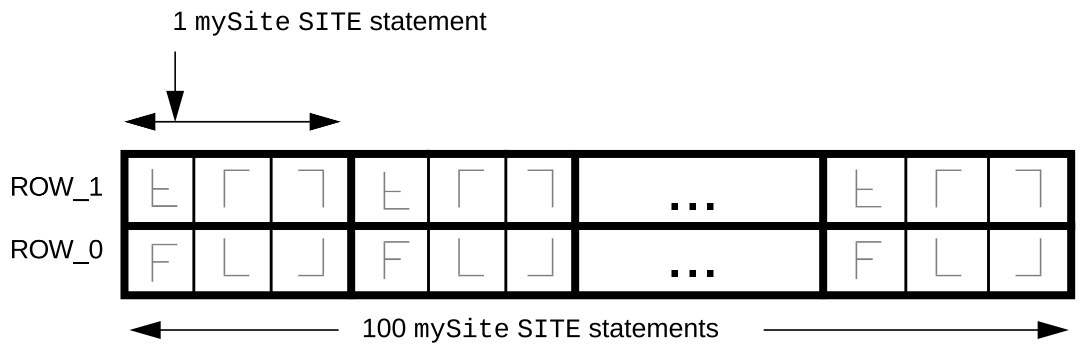
Example 1-29 Site Row Pattern Statement

The following example defines three sites: `Fsite`; `Lsite`; and `mySite`, which consists of a pattern of `Fsite` and `Lsite` sites:

```
SITE Fsite
    CLASS CORE ;
    SIZE 4.0 BY 7.0 ; #4.0 um width, 7.0 um height
END Fsite
SITE Lsite
    CLASS CORE ;
    SIZE 6.0 BY 7.0 ; #6.0 um width, 7.0 um height
END Lsite
SITE mySite
    ROWPATTERN Fsite N Lsite N Lsite FS ; #Pattern of F + L + flipped L
    SIZE 16.0 BY 7.0 ;                    #Width = width(F + L + L)
END mySite
```

Figure 1-22 on page 103 illustrates some DEF rows made up of `mySite` sites.

Figure 1-22



```
ROW ROW_0 mySite 1000 1000 N DO 100 BY 1 STEP 1600 0 ;
ROW ROW_1 mySite 1000 1700 FS DO 100 BY 1 STEP 1600 0 ;
```

SITE *siteName*	Specifies the name for the placement site.
SIZE *width* BY *height*	Specifies the dimensions of the site in normal (or north) orientation, in microns.

SYMMETRY {X | Y | R90}

Indicates which site orientations are equivalent. The sites in a given row all have the same orientation as the row. Generally, site symmetry should be used to control the flipping allowed inside the rows. For more information on defining symmetry, see "Defining Symmetry" on page 75.

Possible orientations include:

X	Site is symmetric about the x axis. This means that N and FS sites are equivalent, and FN and S sites are equivalent. A macro with an orientation of N matches N or FS rows.

`Y`	Site is symmetric about the y axis. This means that N and FN sites are equivalent, and FS and S sites are equivalent. A macro with an orientation of N matches N or FN rows.
`X Y`	Site is symmetric about the x and y axis. This means that N, FN, FS, and S sites are equivalent. A macro with orientation N matches N, FN, FS, or S rows.
`R90`	Site is symmetric when rotated 90 degrees. Typically, this value is not used.

Note: Typically, a site for single-height standard cells uses symmetry Y, and a site for double-height standard cells uses symmetry X Y.

Units

```
[UNITS
     [TIME NANOSECONDS convertFactor ;]
     [CAPACITANCE PICOFARADS convertFactor ;]
     [RESISTANCE OHMS convertFactor ;]
     [POWER MILLIWATTS convertFactor ;]
     [CURRENT MILLIAMPS convertFactor ;]
     [VOLTAGE VOLTS convertFactor ;]
     [DATABASE MICRONS LEFconvertFactor ;]
     [FREQUENCY MEGAHERTZ convertFactor ;]
END UNITS]
```

Defines the units of measure in LEF. The values tell you how to interpret the numbers found in the LEF file. Units are fixed with a *`convertFactor`* for all unit types, except database units and capacitance. For more information, see “Convert Factors” on page 106. Currently, other values for *`convertFactor`* appearing in the `UNITS` statement are ignored.

The `UNITS` statement is optional and, when used, must precede the `LAYER` statements.

`CAPACITANCE PICOFARADS` *`convertFactor`*
Interprets one LEF capacitance unit as 1 picofarad.

`CURRENT MILLIAMPS` *`convertFactor`*
Interprets one LEF current unit as 1 milliamp.

`DATABASE MICRONS` *`LEFconvertFactor`*
Interprets one LEF distance unit as multiplied when converted into database units.

If you omit the `DATABASE MICRONS` statement, a default value of 100 is recorded as the *LEFconvertFactor* in the database. In this case, one micron would equal 100 database units.

`FREQUENCY MEGAHERTZ` *`convertFactor`*
Interprets one LEF frequency unit as 1 megahertz.

`POWER MILLIWATTS` *`convertFactor`*
Interprets one LEF power unit as 1 milliwatt.

`RESISTANCE OHMS` *`convertFactor`*
Interprets one LEF resistance unit as 1 ohm.

`TIME NANOSECONDS` *`convertFactor`*
Interprets one LEF time unit as 1 nanosecond.

`VOLTAGE VOLTS` *`convertFactor`*
Interprets one LEF voltage unit as 1 volt.

Database Units Information

Database precision is relative to Standard International (SI) units. LEF values are converted to integer values in the library database as follows.

SI unit	Database precision
1 nanosecond	= 1,000 DBUs
1 picofarad	= 1,000,000 DBUs
1 ohm	= 10,000 DBUs
1 milliwatt	= 10,000 DBUs
1 milliamp	= 10,000 DBUs
1 volt	= 1,000 DBUs

Convert Factors

LEF supports values of 100, 200, 1000, 2000, 10,000, and 20,000 for *`LEFconvertFactor`*. The following table illustrates the conversion of LEF distance units into database units.

LEFconvertFactor	LEF	Database Units
100	1 micron	100
200	1 micron	200
1000	1 micron	1000
2000	1 micron	2000
10,000	1 micron	10,000
20,000	1 micron	20,000

The DEF database precision cannot be more precise than the LEF database precision. This means the DEF convert factor must always be less than or equal to the LEF convert factor. The following table shows the valid pairings of the LEF convert factor and the corresponding DEF convert factor.

LEFconvertFactor	Legal DEFconvertFactors
100	100
200	100, 200
1000	100, 200, 1000
2000	100, 200, 1000, 2000
10,000	100, 200, 1000, 2000, 10,000
20,000	100, 200, 1000, 2000, 10,000, 20,000

An incremental LEF should have the same value as a previous LEF. An error message warns you if an incremental LEF has a different value than what is recorded in the database.

Use Min Spacing

```
[USEMINSPACING OBS { ON | OFF } ;]
```

Defines how minimum spacing is calculated for obstruction (blockage) geometries.

`OBS {ON | OFF}`	Specifies how to calculate minimum spacing for obstruction geometries. *Default:* `OFF`	
	`OFF`	Spacing is the range spacing that corresponds to the minimum width (in x and y directions) of the obstruction.
	`ON`	Spacing is the minimum layer spacing.

Version

```
VERSION number ;
```

Specifies which version of the LEF syntax is being used. *`number`* is a string of the form *`major`*`.`*`minor`*`[.`*`subMinor`*`]`, such as `5.6`.

Note: Many applications default to the latest version of LEF/DEF supported by the application (which depends on how old the application is). The latest version as described by this document is 5.6. However, a default value of `5.6` is not formally part of the language definition; therefore, you cannot be sure that all applications use this default value. Also, because the default value varies with the latest version, you should not depend on this.

Via

```
VIA viaName [DEFAULT]
     { VIARULE viaRuleName ;
          CUTSIZE xSize ySize ;
          LAYERS botMetalLayer cutLayer topMetalLayer ;
          CUTSPACING xCutSpacing yCutSpacing ;
          ENCLOSURE xBotEnc yBotEnc xTopEnc yTopEnc ;
          [ROWCOL numCutRows numCutCols ;
          [ORIGIN xOffset yOffset ;]
          [OFFSET xBotOffset yBotOffset xTopOffset yTopOffset ;]
          [PATTERN cutPattern ;]
     | [RESISTANCE resistValue ;]
       {LAYER layerName ;
          { RECT pt pt ;
          | POLYGON pt pt pt ...;} ...
       } ...
     }
     [PROPERTY propName propVal ;] ...
END viaName
```

Defines two types of vias: fixed vias and generated vias.

A fixed via is defined using rectangles or polygons, and does not use a `VIARULE`. The fixed via name must mean the same via in all associated LEF and DEF files.

A generated via is defined using `VIARULE` parameters to indicate that it was derived from a `VIARULE GENERATE` statement. For a generated via, the via name is only used locally inside this LEF file. The geometry and parameters are maintained, but the name can be freely changed by applications that use this via when writing out LEF and DEF files. For example, large blocks that include generated vias as part of the LEF `MACRO PIN` statement can define generated vias inside the same LEF file without concern about via name collisions in other LEF files.

Note: Use the `VIARULE GENERATE` statement to define special wiring.

`CUTSIZE` *`xSize ySize`*

Specifies the required width (*`xSize`*) and height (*`ySize`*) of the cut layer rectangles.
Type: Float, specified in microns

`CUTSPACING` *`xCutSpacing yCutSpacing`*

Specifies the required x and y spacing between cuts. The spacing is measured from one cut edge to the next cut edge.
Type: Float, specified in microns

`DEFAULT`

Identifies the via as the default via between the defined layers. Default vias are used for default routing by the signal routers.

If you define more than one default via for a layer pair, the router chooses which via to use. Some routers limit the number of vias for each layer pair to 30. You must define default vias between *`metal1`* and masterslice layers if there are pins on the masterslice layers.

Default vias use exactly three layers: a cut layer, and two layers that touch the cut layer (routing or masterslice). The cut layer rectangle must be between the two routing or masterslice layer rectangles.

`ENCLOSURE` *`xBotEnc yBotEnc xTopEnc yTopEnc`*

Specifies the required x and y enclosure values for the bottom and top metal layers. The enclosure measures the distance from the cut array edge to the metal edge that encloses the cut array.
Type: Float, specified in microns

Note: It is legal to specify a negative number, as long as the resulting metal size is positive.

`LAYER` *`layerName`*

Specifies the layer on which to create the rectangles that make up the via. Vias have exactly three layers used: a cut layer, and two layers that touch the cut layer (routing or masterslice). There should be at least one `RECT` or `POLYGON` on each of the three layers.

`LAYERS` *`botMetalLayer cutLayer topMetalLayer`*

Specifies the required names of the bottom routing layer, cut layer, and top routing layer. These layer names must be previously defined in layer definitions, and must match the layer names defined in the specified LEF *`viaRuleName`*.

`OFFSET` *`xBotOffset yBotOffset xTopOffset yTopOffset`*

Specifies the x and y offset for the bottom and top metal layers. By default, the `0,0` origin of the via is the center of the cut array, and the enclosing metal rectangles. These values allow each metal layer to be offset independently. After the non-shifted via is computed, the metal layer rectangles are offset by adding the appropriate values—the x/y *`BotOffset`* values to the metal layer below the cut layer, and the x/ y *`TopOffset`* values to the metal layer above the cut layer. These offsets are in addition to any offset caused by the `ORIGIN` values.
Default: `0`, for all values
Type: Float, specified in microns

`ORIGIN` *`xOffset yOffset`*

Specifies the x and y offset for all of the via shapes. By default, the `0,0` origin of the via is the center of the cut array, and the enclosing metal rectangles. After the non-shifted via is computed, all cut and metal rectangles are offset by adding these values.
Default: `0`, for both values
Type: Float, specified in microns

`PATTERN` *`cutPattern`*

Specifies the cut pattern encoded as an ASCII string. This parameter is only required when some of the cuts are missing from the array of cuts, and defaults to “all cuts are present,” if not specified.

For information on and examples of via cut patterns, see “Creating Via Cut Patterns” on page 212.

The *`cutPattern`* syntax uses "_" as a separator, and is defined as follows:

```
numRows_rowDefinition
  [_numRows_rowDefinition] ...
```

`numRows`	Specifies a hexadecimal number that indicates how many times to repeat the following row definition. This number can be more than one digit.
`rowDefinition`	Defines one row of cuts, from left to right.

The *`rowDefinition`* syntax is defined as follows:

```
{[RrepeatNumber]hexDigitCutPattern} ...
```

`hexDigitCutPattern`	Specifies a single hexadecimal digit that encodes a 4-bit binary value, in which `1` indicates a cut is present, and `0` indicates a cut is not present.
`repeatNumber`	Specifies a single hexadecimal digit that indicates how many times to repeat *`hexDigitCutPattern`*.

`POLYGON` *`pt pt pt`* Specifies a sequence of at least three points to generate a polygon geometry. The polygon edges must be parallel to the x axis, to the y axis, or at a 45-degree angle. Each `POLYGON` keyword defines a polygon generated by connecting each successive point, and then connecting the first and last points. The *`pt`* syntax corresponds to an x y coordinate pair, such as `-0.2 1.0`.
Type: Float, specified in microns

Example 1-30 Via Polygon

The following via definition creates a polygon geometry used by X-routing applications:

```
VIA myVia23
    LAYER metal2 ;
    POLYGON -2.1 -1.0 -0.2 1.0 2.1 1.0 0.2 -1.0 ;
    LAYER cut23 ;
    RECT -0.4 -0.4 0.4 0.4 ;
    LAYER metal3 ;
    POLYGON -0.2 -1.0 -2.1 1.0 0.2 1.0 2.1 -1.0 ;
END myVia23
```

`PROPERTY` *`propName propVal`*

Specifies a numerical or string value for a via property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`RECT` *`pt pt`*

Specify the corners of a rectangular shape in the via. The *`pt`* syntax corresponds to an x y coordinate pair, such as `-0.4 -4.0`. For vias used only in macros or pins, reference locations and rectangle coordinates must be consistent.
Type: Float, specified in microns

`RESISTANCE` *`resistValue`*

Specifies the lumped resistance for the via. This is not a resistance per via-cut value; it is the total resistance of the via. By default, via resistance is computed from the via `LAYER RESISTANCE` value; however, you can override that value with this value. *`resistValue`* is ignored if a via rule is specified, because only the `VIARULE` definition or a cut layer `RESISTANCE` value gives the resistance for generated vias.
Type: Float, specified in ohms

Note: A `RESISTANCE` value attached to an individual via is no longer recommended.

`ROWCOL` *`numCutRows numCutCols`*

Specifies the number of cut rows and columns that make up the via array.
Default: `1`, for both values
Type: Positive integer, for both values

`viaName`

Specifies the name for the via.

`VIARULE` *`viaRulename`*

Specifies the name of the LEF `VIARULE` that produced this via. This indicates that the via is the result of automatic via generation, and that the via name is only used locally inside this LEF file. The geometry and parameters are maintained, but the name can be freely changed by applications that use this via when writing out LEF and DEF files.

viaRuleName must be specified before you define any of the other parameters, and must refer to a previously defined `VIARULE GENERATE` rule name. It cannot refer to a `VIARULE` without a `GENERATE` keyword.

Specifying the reserved via rule name of `DEFAULT` indicates that the via should use a previously defined `VIARULE GENERATE` rule with the `DEFAULT` keyword that exists for this routing-cut-routing layer combination. This makes it possible for an IP block user to use existing via rules from the normal LEF technology section instead of requiring it to locally create its own via rules for just one LEF file.

Example 1-31 Generated Via Rule

The following via definition defines a generated via that is used only in this LEF file.

```
VIA myBlockVia
    VIARULE DEFAULT ;                  #Use existing VIARULE GENERATE rule with
                                        #the DEFAULT keyword
        CUTSIZE 0.1 0.1 ;               #Cut is 0.1 x 0.1 um
        LAYERS metal1 via12 metal2 ;    #Bottom metal, cut, and top metal layers
        CUTSPACING 0.1 0.1 ;            #Space between cut edges is 0.1 um
        ENCLOSURE 0.05 0.01 0.01 0.05 ; #metal1 enclosure is 0.05 in x, 0.01 in y
                                        #metal2 enclosure is 0.01 in x, 0.05 in y
        ROWCOL 1 2 ;                    #1 row, 2 columns = 2 cuts
END myBlockVia
```

Via Rule

```
VIARULE viaRuleName
     LAYER layerName ;
        DIRECTION {HORIZONTAL | VERTICAL} ;
        [WIDTH minWidth TO maxWidth ;]
     LAYER layerName ;
        DIRECTION {HORIZONTAL | VERTICAL} ;
        [WIDTH minWidth TO maxWidth ;]
     {VIA viaName ;} ...
     [PROPERTY propName propVal ;] ...

END viaRuleName
```

Defines which vias to use at the intersection of special wires of the same net.

Note: You should only use `VIARULE GENERATE` statements to create a via for the intersection of two special wires. In earlier versions of LEF, `VIARULE GENERATE` was not complete enough to cover all situations. In those cases, a fixed `VIARULE` (without a `GENERATE` keyword) was sometimes used. However, fixed `VIARULE` statements are no longer recommended.

`DIRECTION {HORIZONTAL | VERTICAL}`

Specifies the wire direction. If you specify a `WIDTH` range, the rule applies to wires of the specified `DIRECTION` that fall within the range. Otherwise, the rule applies to all wires of the specified `DIRECTION` on the layer.

`LAYER` *`layerName`*

Specifies the routing layers for the top or bottom of the via.

`PROPERTY` *`propName propVal`*

Specifies a numerical or string value for a via rules property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`VIA` *`viaName`*

Specifies a previously defined via to test for the current via rule. The first via in the list that can be placed at the location without design rule violations is selected. The vias must all have exactly three layers in them. The three layers must include the same routing layers as listed in the `LAYER` statements of the `VIARULE`, and a cut layer that is between the two routing layers.

`VIARULE` *`viaRuleName`*

Specifies the name to identify the via rule.

`WIDTH` *`minWidth`* `TO` *`maxWidth`*

Specifies a wire width range. If the widths of two intersecting special wires fall within the wire width range, the `VIARULE` is used. To fall within the range, the widths must be greater than or equal to *`minWidth`* and less than or equal to *`maxWidth`*.

Note: `WIDTH` is defined by wire direction, not by layer. If you specify a `WIDTH` range, the rule applies to wires of the specified `DIRECTION` that fall within the range.

Example 1-32 Via Rule Statement

In the following example, whenever a *metal1* wire with a width between 0.5 and 1.0 intersects a *metal2* wire with a width between 1.0 and 2.0, the via generation code attempts to put a *via12_1* at the intersection first. If the *via12_1* causes a DRC violation, a *via12_2* is then tried. If both fail, the default behavior from a `VIARULE GENERATE` statement for *metal1* and *metal2* is used.

```
VIARULE viaRule1
    LAYER metal1 ;
        WIDTH 0.5 TO 1.0 ;
    LAYER metal2 ;
        WIDTH 1.0 TO 2.0 ;
    VIA via12_1 ;
    VIA via12_2 ;
END viaRule1
```

Via Rule Generate

```
VIARULE viaRuleName GENERATE [DEFAULT]
     LAYER routingLayerName ;
        ENCLOSURE overhang1 overhang2 ;
        [WIDTH minWidth TO maxWidth ;]
     LAYER routingLayerName ;
        ENCLOSURE overhang1 overhang2 ;
        [WIDTH minWidth TO maxWidth ;]
     LAYER cutLayerName ;
        RECT pt pt ;
        SPACING xSpacing BY ySpacing ;
        [RESISTANCE resistancePerCut ;]

END viaRuleName
```

Defines formulas for generating via arrays. You can use the `VIARULE GENERATE` statement to cover special wiring that is not explicitly defined in the `VIARULE` statement.

Rather than specifying a list of vias for the situation, you can create a formula to specify how to generate the cut layer geometries.

Note: Any vias created automatically from a `VIARULE GENERATE` rule that appear in the DEF `NETS` or `SPECIALNETS` sections must also appear in the DEF `VIA` section.

`DEFAULT`	Specifies that the via rule can be used to generate vias for the default routing rule. There can only be one `VIARULE GENERATE DEFAULT` for a given routing-cut-routing layer combination.

Example 1-33 Via Rule Generate Default

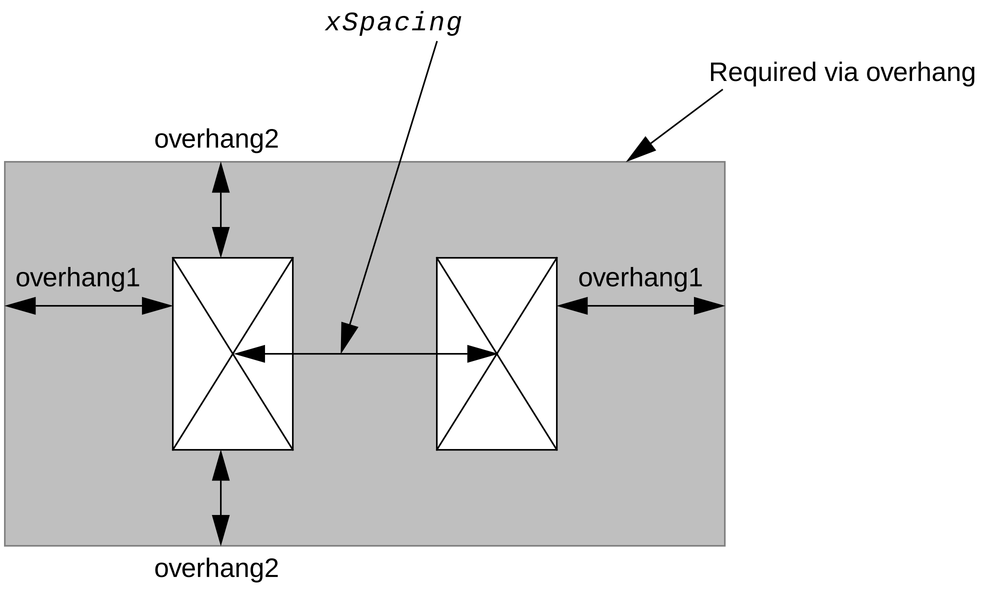
The following example defines a rule for generating vias for the default routing rule:

```
VIARULE via12 GENERATE DEFAULT
    LAYER m1 ;
    ENCLOSURE 0.03 0.01 ; #2 sides need >= 0.03, 2 other sides need >= 0.01
    LAYER m2 ;
    ENCLOSURE 0.05 0.01 ; #2 sides need >= 0.05, 2 other sides need >= 0.01
    LAYER cut12 ;
    RECT -0.1 -0.1 0.1 0.1 ; # cut is .20 by .20
    SPACING 0.40 BY 0.40 ;   #center-to-center spacing
    RESISTANCE 20 ;          #ohms per cut
END via12
```

`ENCLOSURE` *`overhang1 overhang2`*

Specifies that the via must be covered by metal on two opposite sides by at least *`overhang1`*, and on the other two sides by at least *`overhang2`* (see Figure 1-23 on page 115). The via generation code then chooses the direction of overhang that best maximizes the number of cuts that can fit in the via.
Type: Float, specified in microns

Figure 1-23 Overhang



Example 1-34 Via Rule Generate Enclosure

The following example describes a formula for generating via cuts:

```
VIARULE via12 GENERATE
    LAYER m1 ;
        ENCLOSURE 0.05 0.01 ; #2 sides must be >=0.05, 2 other sides must be >=0.01
        WIDTH 0.2 TO 100.0 ;  #for m1, between 0.2 to 100 microns wide
    LAYER m2 ;
        ENCLOSURE 0.05 0.01 ; #2 sides must be >=0.05, 2 other sides must be >=0.01
        WIDTH 0.2 TO 100.0 ;  #for m2, between 0.2 to 100 microns wide
    LAYER cut12
    RECT -0.07 -0.07 0.07 0.07 ; #cut is .14 by .14
    SPACING 0.30 BY 0.30 ;       #center-to-center spacing
END via12
```

The cut layer `SPACING ADJACENTCUTS` statement can override the `VIARULE` cut layer `SPACING` statements. For example, assume the following cut layer information is also defined in the LEF file:

```
LAYER cut12
    ...
    SPACING 0.20 ADJACENTCUTS 3 WITHIN 0.22 ;
    ...
```

The 0.20 µm edge-to-edge spacing in the `ADJACENTCUTS` statement is larger than the `VIARULE GENERATE` example spacing of 0.16 (0.30 – 0.14). Whenever the `VIARULE GENERATE` rule creates a via that is larger that 2x2 cuts (that is, 2x3, 3x2, 3x3 and so on), the 0.20 spacing from the `ADJACENTCUTS` statement is used instead.

Note: The spacing in `VIARULE GENERATE` is center-to-center spacing, whereas the spacing in `ADJACENTCUTS` is edge-to-edge.

`GENERATE`	Defines a formula for generating the appropriate via.
`LAYER` *`cutLayerName`*	Specifies the cut layer for the generated via.
`LAYER` *`routingLayerName`*	Specifies the routing layers for the top and bottom of the via.
`RECT` *`pt pt`*	Specifies the location of the lower left contact cut rectangle.
`RESISTANCE` *`resistancePerCut`*	Specifies the resistance of the cut layer, given as the resistance per contact cut.

Default: The resistance value in the `LAYER` (Cut) statement
Type: Float

`SPACING` *`xSpacing`* `BY` *`ySpacing`*

Defines center-to-center spacing in the x and y dimensions to create an array of contact cuts.The number of cuts of an array in each direction is the most that can fit within the bounds of the intersection formed by the two special wires. Cuts are only generated where they do not violate stacked or adjacent via design rules.

Note: This value can be overridden by the `SPACING ADJACENTCUTS` value in the cut layer statement.

`VIARULE` *`viaRuleName`*

Specifies the name for the rule.

The name `DEFAULT` is reserved and should not be used for any via rule name. In the LEF and DEF `VIA` definitions that use generated via parameters, the reserved `DEFAULT` name indicates the via rule with the `DEFAULT` keyword.

`WIDTH` *`minWidth`* `TO` *`maxWidth`*

Specifies a wire width range within which the wire must fall in order for the rule to apply. That is, the wire width must be greater than or equal to *`minWidth`* and less than or equal to *`maxWidth`*.

2

ALIAS Statements

This chapter contains information about the following topics.

ALIAS Statements

You can use alias statements in LEF and DEF files to define commands or parameters associated with the library or design. An alias statement can appear anywhere in a LEF or DEF file as follows:

```
&ALIAS  &&aliasName  =  aliasDefinition   &ENDALIAS
```

`&ALIAS` and `&ENDALIAS` are both reserved keywords and are not case sensitive. An alias statement has the following requirements:

- `&ALIAS` must be the first token in the line in which it appears.
- *aliasName* is string name and must appear on the same line as *&ALIAS.* It is case sensitive based on the value of `NAMESCASENSITIVE` in the LEF input, or the value of `Input.Lef.Names.Case.Sensitive`*.*
- *aliasName* cannot contain any of the following special characters: #, space, tab, or control characters.
- `&ENDALIAS` must be the last token in the line in which it appears.
- Multiple commands can appear in the alias definition, separated by semicolons. However, the last command must not be terminated by a semicolon.

ALIAS Definition

The alias name (*aliasName*) is an identifier for the associated alias definition (*aliasDefinition*). The data reader stores the alias definition in the database. If the associated alias name already exists in the database, a warning is issued and the existing definition is replaced.

Alias definitions are text strings with the following properties:

- *aliasDefinition* is any text excluding "`&ENDALIAS`".
- All EOL, space, and tab characters are preserved.
- *aliasDefinition* text can expand to multiple lines.

ALIAS Examples

The following examples include legal and illegal alias statements:

- The following statement is legal.

  ```
  &ALIAS &&MAC = SROUTE ADDCELL AREA &&CORE &ENDALIAS
  ```

- The following statement is illegal because `MAC` does not start with "&&".

  ```
  &ALIAS MAC = SROUTE AREA &&CORE &ENDALIAS
  ```

- The following statement is illegal because `&ALIAS` is not the first token in this line.

  ```
  ( 100 200 ) &ALIAS &&MAC = SROUTE AREA &&CORE &ENDALIAS
  ```

- The following statement is legal. It contains multiple commands; the last command is not terminated by a semicolon.

  ```
  $ALIAS $$ = INPUT LEF myfile.txt;
  VERIFY LIBRARY
  ENDALIAS
  ```

The following examples show legal and illegal alias names:

- "`Engineer_change`" is a legal alias name.

  ```
  &&Engineer_change
  ```

- "`&Version&History&&`" is a legal alias name.

  ```
  &&&Version&History&&
  ```

- "`design history`" is an illegal alias name. It contains a space character and is considered as two tokens: an *aliasName* token "`&&design`," and a non-*aliasName* token "`history`".

  ```
  &&design history
  ```

- "`someName#IO-pin-Num`" is an illegal alias name. It contains a "#" character and is translated as one *aliasName* token "`&&someName`". The "#" is considered a comment character.

  ```
  &&someName#IO-pin-Num
  ```

ALIAS Expansion

Alias expansion is the reverse operation of alias definition. The following is the syntax for alias expansion.

`&&`*`aliasName`*

where *aliasName* is any name previously defined by an alias statement. If an *aliasName* does not exist in the database, no substitution occurs.

You use aliases as string expansion parameters for LEF or DEF files. An alias can substitute for any token of a LEF or DEF file.

3

Working with LEF

This chapter contains information about the following topics.

Incremental LEF

`INPUT LEF` can add new data to the current database, providing an incremental LEF capability. Although it is possible to put an entire LEF library in one file, some systems require that you put certain data in separate files.

This feature also is useful, for example, when combined with the `INPUT GDSII` command, to extract geometric data from a GDSII-format file and add the data to the database.

When using `INPUT LEF` on a database that has been modified previously, save the previous version before invoking `INPUT LEF`. This provides a backup in case the library information has problems and the database gets corrupted or lost.

Important

The original LEF file, created with `FINPUT LEF` (or with `INPUT LEF` when no database is loaded), must contain all the layers.

Adding Objects to the Library

`INPUT LEF` can add the following objects to the database:

- New via
- New via rule
- Samenet spacings (if none have been specified previously)
- New macro

If geometries have not been specified for an existing via, `INPUT LEF` can add layers and associated rectangle geometries. If not specified previously for a macro, `INPUT LEF` can add the following:

- `FOREIGN` statement
- EEQ
- LEQ
- Size
- Overlap geometries
- Obstruction geometries

If not previously specified for an existing macro pin, `INPUT LEF` can add the following:

- Mustjoins
- Ports and geometries

The database created by `INPUT LEF` can contain a partial library. Run `VERIFY LIBRARY` before proceeding.

If new geometries are added to a routed database, run `VERIFY GEOMETRY` and `VERIFY CONNECTIVITY` to identify new violations.

Important

When defining a pin with no port geometries with the intent of incrementally adding them, *do not* include an empty `PORT` statement as shown below.

```
MACRO abc
     ...
     PIN a
           ...
           PORT  # dummy pin-port, do not
           END   # include these two lines
     END a
     ...
```

Error Checking

To help develop, test, and debug generic libraries and parametric macros, LEF and DEF have a user-defined error checking facility. This facility consists of seven utilities that you can use from within a LEF or DEF file during the scanning phase of LEF/DEF readers. These utilities have the following features:

- A message facility that writes to one or more text files during LEF or DEF input
- An error handling facility that logs user detected warnings, errors, and fatal errors

The error checking utilities have the following syntax:

```
&CREATEFILE &fileAlias =
    { stringExpression
    | stringIF-ELSEexpression } ;

&OPENFILE &fileAlias ;

&CLOSEFILE &fileAlias ;

&MESSAGE
    {&fileAlias | &MSGWINDOW} = message;

&WARNING
    {&fileAlias | &MSGWINDOW} = message ;

&ERROR
    {&fileAlias | &MSGWINDOW} = message ;

&FATALERROR
    {&fileAlias | &MSGWINDOW} = message;

message =
    { &fileAlias | stringExpression
    | stringIF-ELSEexpression
    | stringIFexpression }
```

Message Facility

The message facility outputs user-defined messages during the scanning phase of LEF and DEF input. These messages can be directed to the message window.

&CREATEFILE

The `&CREATEFILE` utility first assigns a token (*&fileAlias*) to represent a named file. The file name is derived from a previously defined string, a quoted string, or an `IF-ELSE` expression that evaluates to a string. The following example illustrates these three cases.

```
&DEFINES &messagefile = "demo1.messages" ;
&CREATEFILE &outfile = &messagefile ;
&CREATEFILE &msgs =
        "/usr/asics/cmos/fif4/errors.txt" ;
```

```
&CREATEFILE &messages =
        IF &errortrap
            THEN "errs.txt"
            ELSE "/dev/null" ;
```

The derived file name must be a legal file name in the host environment. The default directory is the current working directory. The file names are case sensitive.

`&CREATEFILE` creates an empty file with the given name and opens the file. If the token is already bound to another open file, a warning is issued, the file is closed, and the new file is opened. If the file already exists, the version number is incremented.

&CLOSEFILE and &OPENFILE

The `&CLOSEFILE` utility closes the file bound to a given token; `&OPENFILE` opens the file bound to a given token. `&CLOSEFILE` and `&OPENFILE` control the number of open files. Each operating system has a limit for the number of open files. Therefore, `&CLOSEFILE` might be needed to free up extra file descriptors.

Files are closed in the following ways.

- All user files are closed at the end of the scanning phase of the LEF and DEF readers.
- All user files are closed if the scanning phase aborts.
- If `&CREATEFILE` is invoked with a token that is already bound to an open file, that file is closed before opening the new file.

&MESSAGE

The `&MESSAGE` utility appends text to the file represented by the *&fileAlias* token, or to the message window if `&MSGWINDOW` is specified.

`&MSGWINDOW` is a special file alias that is not created, opened, or closed. The assigned expression (right side of the statement) can be one of the following:

`&fileAlias`	Must correspond to a valid file that has been successfully opened. The contents of the file are appended to the target file (or message window).
`stringExpression`	Either a string or a string token. For example:

```
&DEFINES &romword16 =
    "ROM word size = 16 bits" ;
&MESSAGE &mesgs = "ROM size = 256" ;
&MESSAGE &mesgs = &romword16 ;
```

stringIF-ELSEexpression

String `IF-ELSE` expressions evaluate a Boolean expression and then branch to string values, for example:

```
&&MESSAGE &mesgs =
    IF (&&c_flag = 0)
        THEN "FLAG C set to 0"
    ELSE IF ( &&c_flag = 1 )
        THEN "FLAG C set to 1"
    ELSE "FLAG C set to 2" ;
```

As shown in this example, `IF-ELSE` expressions can be nested.

stringIFexpression

A string `IF` expression is an `IF-ELSE` expression without the `ELSE` phrase. The Boolean expression is evaluated, and if true, the `THEN` string is sent to the target file; if false, no string is sent, for example,

```
&MESSAGE &mesgs =
    IF ( &&buf = "INV_BIG" )
        THEN "INV_BIG buffers" ;
```

Neither the file alias token nor `&MSGWINDOW` can be part of the assigned expression.

Error-Checking Facility

In addition to the message facility, you have partial control of the error checking facility of the LEF and DEF readers. When scanning LEF or DEF input, the readers record warnings, errors, and fatal errors. At the end of the scan, the total number of each is sent to the message window before proceeding with the reader phase.

If a fatal error is detected, input is aborted after the scanning phase.

With the user interface to the error checking facility, the LEF and DEF files can include custom error checking. User detected warnings, errors, and fatal errors, can be logged, thereby incrementing the DEF/LEF reader’s warning, error, and fatal error counts.

A user-detected fatal error terminates input just as with the resident error checking facility. In addition, the user defined error checking facility utilities can send message strings to the message window.

&WARNING, &ERROR, and &FATALERROR

The `&WARNING`, `&ERROR`, and `&FATALERROR` utilities use the same syntax as the `&MESSAGE` utility. These utilities can send message strings to files and to the message window in the same manner as `&MESSAGE`. In addition, when the assigned expression is a string `IF` expression, or a string `IF-ELSE` expression, then the associated counter (warnings, errors, or fatal errors) is incremented by `1` if any `IF` condition evaluates to true.

4

DEF Syntax

This chapter contains information about the following topics:

About Design Exchange Format Files

A Design Exchange Format (DEF) file contains the design-specific information of a circuit and is a representation of the design at any point during the layout process. The DEF file is an ASCII representation using the syntax conventions described in "Typographic and Syntax Conventions" on page 7.

DEF conveys logical design data to, and physical design data from, place-and-route tools. Logical design data can include internal connectivity (represented by a netlist), grouping information, and physical constraints. Physical data includes placement locations and orientations, routing geometry data, and logical design changes for backannotation. Place-and-route tools also can read physical design data, for example, to perform ECO changes.

For standard-cell-based/ASIC flow tools, floorplanning is part of the design flow. You typically use the various floorplanning commands to interactively create a floorplan. This data then becomes part of the physical data output for the design using the `ROWS`, `TRACKS`, `GCELLGRID`, and `DIEAREA` statements. You also can manually enter this data into DEF to create the floorplan.

It is legal for a DEF file to contain only floorplanning information, such as `ROWS`. In many cases, the DEF netlist information is in a separate format, such as Verilog, or in a separate

DEF file. It is also common to have a DEF file that only contains a `COMPONENTS` section to pass placement information.

General Rules

Note the following information about creating DEF files:

- Lines in the DEF file are limited to 2,048 characters (extra characters are truncated on input).
- Net names and cell names also are limited to 2,048 characters.
- DEF statements end with a semicolon (`;`). You *must* leave a space before the semicolon.
- Each section can be specified only once. Sections end with `END` *`SECTION`*.
- You must define all objects before you reference them except for the `+ ORIGINAL` argument in the `NETS` section.
- No regular expressions or wildcard characters are recognized except for `( * `*`pinName`*` )` in the `SPECIALNETS` section.

Order of DEF Statements

Standard DEF files can contain the following statements and sections. You can define the statements and sections in any order; however, data must be defined before it is used. For example, you must specify the `UNITS` statement before any statements that use values dependent on `UNITS` values, and `VIAS` statements must be defined before statements that use via names. If you specify statements and sections in the following order, all data is defined before being used.

```
[ VERSION statement ]
[ DIVIDERCHAR statement ]
[ BUSBITCHARS statement ]
DESIGN statement
[ TECHNOLOGY statement ]
[ UNITS statement ]
[ HISTORY statement ] ...
[ PROPERTYDEFINITIONS section ]
[ DIEAREA statement ]
[ ROWS statement ] ...
[ TRACKS statement ] ...
[ GCELLGRID statement ] ...
[ VIAS statement ]
[ STYLES statement ]
```

```
     [ NONDEFAULTRULES statement ]
     [ REGIONS statement ]
     [ COMPONENTS section ]
     [ PINS section ]
     [ PINPROPERTIES section ]
     [ BLOCKAGES section ]
     [ SLOTS section ]
     [ FILLS section ]
     [ SPECIALNETS section ]
     [ NETS section ]
     [ SCANCHAINS section ]
     [ GROUPS section ]
     [ BEGINEXT section ] ...
     END DESIGN statement
```

DEF Statement Definitions

The following definitions describe the syntax arguments for the statements and sections that make up a DEF file. The statements and sections are listed in alphabetical order, *not* in the order they must appear in a DEF file. For the correct order, see Order of DEF Statements on page 131.

Blockages

```
[BLOCKAGES numBlockages ;
     [- LAYER layerName
          [+ COMPONENT compName | + SLOTS | + FILLS | + PUSHDOWN]
          [+ SPACING minSpacing | + DESIGNRULEWIDTH effectiveWidth]
               {RECT pt pt | POLYGON pt pt pt ...} ...
     ;] ...
     [- PLACEMENT
          [+ COMPONENT compName | + PUSHDOWN]
               {RECT pt pt} ...
     ;] ...
END BLOCKAGES]
```

Defines placement and routing blockages in the design. You can define simple blockages (blockages specified for an area), or blockages that are associated with specific instances (components). You can only associate blockages with placed instances. If you move the instance, its blockage moves with it.

`COMPONENT` *`compName`*	Specifies a component with which to associate a blockage. Specify with `LAYER` *`layerName`* to create a routing blockage associated with a component. Specify with `PLACEMENT` to create a placement blockage associated with a component.

`DESIGNRULEWIDTH` *`effectiveWidth`*	Specifies that the blockage has a width of *`effectiveWidth`* for the purposes of spacing calculations. If you specify `DESIGNRULEWIDTH`, you cannot specify `SPACING`. *Type:* DEF database units
`FILLS`	Creates a blockage on the specified layer where metal fills cannot be placed.
`LAYER` *`layerName`*	Specifies the cut layer or routing layer on which to create a blockage. **Note:** Placing vias using the cut layer where a cut layer obstruction (`OBS`) or blockage exists will cause a violation.
`numBlockages`	Specifies the number of blockages in the design specified in the `BLOCKAGES` section.
`PLACEMENT`	Creates a placement blockage. You can create a simple placement blockage, or a placement blockage attached to a specific component.
`POLYGON` *`pt pt pt`*	Specifies a sequence of at least three points to generate a polygon geometry. The polygon edges must be parallel to the x axis, the y axis, or at a 45-degree angle. Each `POLYGON` statement defines a polygon generated by connecting each successive point, and then the first and last points. The *`pt`* syntax corresponds to a coordinate pair, such as *x y*. Specify an asterisk (`*`) to repeat the same value as the previous *x* or *y* value from the last point.
`PUSHDOWN`	Specifies that the blockage was pushed down into the block from the top level of the design.
`RECT` *`pt pt`*	Specifies the coordinates of the blockage geometry. The coordinates you specify are absolute. If you associate a blockage with a component, the coordinates are not relative to the component's origin.
`SLOTS`	Creates a blockage on the specified layer where slots cannot be placed.

`SPACING` *`minSpacing`*	Specifies the minimum spacing allowed between the blockage and any other routing shape. If you specify `SPACING`, you cannot specify `DESIGNRULEWIDTH`. *Type:* Integer, specified in DEF database units

Example 4-1 Blockages Statements

The following `BLOCKAGES` section defines eight blockages in the following order: two *metal2* routing blockages, a pushed down routing blockage, a routing blockage attached to component `|i4`, a floating placement blockage, a pushed down placement blockage, a placement blockage attached to component `|i3`, and a fill blockage.

```
BLOCKAGES 7 ;
    - LAYER metal1
         RECT ( -300 -310 ) ( 320 330 )
         RECT ( -150 -160 ) ( 170 180 ) ;
    - LAYER metal1 + PUSHDOWN
         RECT ( -150 -160 ) ( 170 180 ) ;
    - LAYER metal1 + COMPONENT |i4
         RECT ( -150 -160 ) ( 170 180 ) ;
    - PLACEMENT
         RECT ( -150 -160 ) ( 170 180 ) ;
    - PLACEMENT + PUSHDOWN
         RECT ( -150 -160 ) ( 170 180 ) ;
    - PLACEMENT + COMPONENT |i3
         RECT ( -150 -160 ) ( 170 180 ) ;
    - LAYER metal1 + FILLS
         RECT ( -160 -170 ) ( 180 190 ) ;
END BLOCKAGES
```

The following `BLOCKAGES` section defines two blockages. One requires minimum spacing of 1000 database units for its rectangle and polygon. The other requires that its rectangle's width be treated as 1000 database units for DRC checking.

```
BLOCKAGES 2 ;
    - LAYER metal1
        + SPACING 1000         #RECT and POLYGON require at least 1000 dbu spacing
        RECT ( -300 -310 ) ( 320 300 )
        POLYGON ( 0 0 ) ( * 100 ) ( 100 * ) ( 200 200 ) ( 200 0 ) ; #Has 45-degree
        edge
    - LAYER metal1
        + DESIGNRULEWIDTH 1000 #Treat the RECT as 1000 dbu wide for DRC checking
        RECT ( -150 -160 ) ( 170 180 ) ;
END BLOCKAGES
```

Bus Bit Characters

```
BUSBITCHARS "delimiterPair" ;
```

Specifies the pair of characters used to specify bus bits when DEF names are mapped to or from other databases. The characters must be enclosed in double quotation marks. For example:

```
     BUSBITCHARS "()" ;
```

If one of the bus bit characters appears in a DEF name as a regular character, you must use a backslash (\) before the character to prevent the DEF reader from interpreting the character as a bus bit delimiter.

If you do not specify the `BUSBITCHARS` statement in your DEF file, the default value is `“[]”`.

Components

```
COMPONENTS numComps ;
     [– compName modelName
         [+ EEQMASTER macroName]
         [+ SOURCE {NETLIST | DIST | USER | TIMING}]
         [+ {FIXED pt orient | COVER pt orient | PLACED pt orient
                | UNPLACED} ]
         [+ HALO left bottom right top]
         [+ WEIGHT weight]
         [+ REGION regionName]
         [+ PROPERTY {propName propVal} ...]...
     ;] ...
END COMPONENTS
```

Defines design components, their location, and associated attributes.

compName modelName	Specifies the component name in the design, which is an instance of *modelName*, the name of a model defined in the library. A *modelName* must be specified with each *compName*.
COVER *pt orient*	Specifies that the component has a location and is a part of a cover macro. A `COVER` component cannot be moved by automatic tools or interactive commands. You must specify the component’s location and its orientation.
EEQMASTER *macroName*	Specifies that the component being defined should be electrically equivalent to the previously defined *macroName*.

`FIXED` *`pt orient`* Specifies that the component has a location and cannot be moved by automatic tools, but can be moved using interactive commands. You must specify the component's location and orientation.

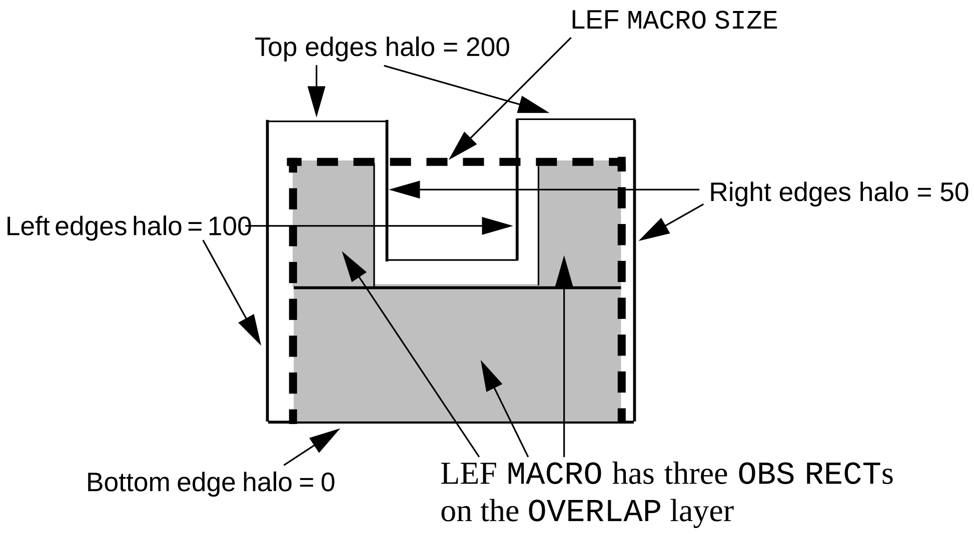
`HALO` *`left bottom right top`*
Specifies a placement blockage around the component. The halo extends from the LEF macro's left edge(s) by *`left`*, from the bottom edge(s) by *`bottom`*, from the right edge(s) by *`right`*, and from the top edge(s) by *`top`*. The LEF macro edges are either defined by the rectangle formed by the `MACRO SIZE` statement, or, if `OVERLAP` obstructions exist (`OBS` shapes on a layer with `TYPE OVERLAP`), the polygon formed by merging the `OVERLAP` shapes.
Type: Integer, specified in DEF database units

Example 4-2 Component Halo

The following statement creates a placement blockage for a "U-shaped" LEF macro, as illustrated in Figure 4-1 on page 136:

```
- i1/i2
    + PLACED ( 0 0 ) N
    + HALO 100 0 50 200 ;
```

Figure 4-1



`*numComps*`	Specifies the number of components defined in the `COMPONENTS` section.
`PLACED *pt orient*`	Specifies that the component has a location, but can be moved using automatic layout tools. You must specify the component's location and orientation.
`PROPERTY *propName propVal*`	Specifies a numerical or string value for a component property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.
`REGION *regionName*`	Specifies a region in which the component must lie. *`regionName`* specifies a region already defined in the `REGIONS` section. If the region is smaller than the bounding rectangle of the component itself, the DEF reader issues an error message and ignores the argument. If the region does not contain a legal location for the component, the component remains unplaced after the placement step.
`SOURCE {NETLIST | DIST | USER | TIMING}`	Specifies the source of the component. Value: Specify one of the following:

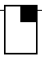
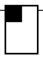
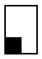
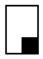
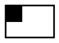
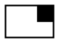
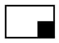
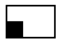
`DIST`	Component is a physical component (that is, it only connects to power or ground nets), such as filler cells, well-taps, and decoupling caps.
`NETLIST`	Component is specified in the original netlist. This is the default value, and is normally not written out in the DEF file.
`TIMING`	Component is a logical rather than physical change to the netlist, and is typically used as a buffer for a clock-tree, or to improve timing on long nets.
`USER`	Component is generated by the user for some user-defined reason.

`UNPLACED`	Specifies that the component does not have a location.

`WEIGHT` *`weight`*	Specifies the weight of the component, which determines whether or not automatic placement attempts to keep the component near the specified location. *`weight`* is only meaningful when the component is placed. All non-zero weights have the same effect during automatic placement. *Default:* `0`

Specifying Orientation

If a component has a location, you must specify its location and orientation. A component can have any of the following orientations:

Value	Definition	Value	Definition
N	North 	FN	Flipped north 
S	South 	FS	Flipped south 
W	West 	FW	Flipped west 
E	East 	FE	Flipped east 

Components are always placed such that the lower left corner of the cell is the origin (0,0) after any orientation. When a component flips about the y axis, it flips about the component center. When a component rotates, the lower left corner of the bounding box of the component's sites remains at the same placement location.

Design

```
DESIGN designName ;
```

Specifies a name for the design. The DEF reader reports a warning if this name is different from that in the database. In case of a conflict, the just specified name overrides the old name.

Die Area

```
[DIEAREA pt pt [pt] ... ;]
```

If two points are defined, specifies two corners of the bounding rectangle for the design. If more than two points are defined, specifies the points of a polygon that forms the die area. The edges of the polygon must be parallel to the x or y axis (45-degree shapes are not allowed), and the last point is connected to the first point. All points are integers, specified as DEF database units.

Geometric shapes (such as blockages, pins, and special net routing) can be outside of the die area, to allow proper modeling of pushed down routing from top-level designs into sub blocks. However, routing tracks should still be inside the die area.

Example 4-3 Die Area Statements

The following statements show various ways to define the die area.

```
DIEAREA (0 0) (100 100) ;                      #A rectangle from 0,0 to 100,100
DIEAREA (0 0) (0 100) (100 100) (100 0) ;      #The same rectangle as a polygon
DIEAREA (0 0) (0 100) (50 100) (50 50) (100 50) (100 0) ; #L-shaped polygon
```

Divider Character

`DIVIDERCHAR "`*`character`*`" ;`

Specifies the character used to express hierarchy when DEF names are mapped to or from other databases. The character must be enclosed in double quotation marks. For example:

```
    DIVIDERCHAR "/" ;
```

If the divider character appears in a DEF name as a regular character, you must use a backslash (\) before the character to prevent the DEF reader from interpreting the character as a hierarchy delimiter.

If you do not specify the `DIVIDERCHAR` statement in your LEF file, the default value is *“/”*.

Extensions

```
[BEGINEXT "tag"
     extensionText
ENDEXT]
```

Adds customized syntax to the DEF file that can be ignored by tools that do not use that syntax. You can also use extensions to add new syntax not yet supported by your version of LEF/DEF, if you are using version 5.1 or later. Add extensions as separate sections.

`extensionText`	Defines the contents of the extension.
`"`*`tag`*`"`	Identifies the extension block. You must enclose *`tag`* in quotes.

Example 4-4 Extension Statement

```
BEGINEXT "1VSI Signature 1.0"
    CREATOR "company name"
    DATE "timestamp"
    REVISION "revision number"
ENDEXT
```

Fills

```
[FILLS numFills ;
     [- LAYER layerName
         {RECT pt pt | POLYGON pt pt pt ...} ...
     ;] ...
END FILLS]
```

Defines the rectangular shapes that represent metal fills in the design. Each fill is defined as an individual rectangle.

`LAYER` *`layerName`*	Specifies the layer on which to create the fill.
`numFills`	Specifies the number of `LAYER` statements in the `FILLS` statement, *not* the number of rectangles.
`POLYGON` *`pt pt pt`*	The polygon edges must be parallel to the x axis, the y axis, or at a 45-degree angle. Each `POLYGON` statement defines a polygon generated by connecting each successive point, and then the first and last points. The *`pt`* syntax corresponds to a coordinate pair, such as *`x y`*. Specify an asterisk (`*`) to repeat the same value as the previous *`x`* or *`y`* value from the last point.
`RECT` *`pt pt`*	Specifies the lower left and upper right corner coordinates of the fill geometry.

Example 4-5 Fills Statements

The following `FILLS` statement defines fills for layers `MET1` and `MET2`:

```
FILLS 2 ;
    - LAYER MET1
        RECT (1000 2000) (1500 4000)
        RECT (2000 2000) (2500 4000)
        RECT (3000 2000) (3500 4000) ;
    - LAYER MET2
        RECT (1000 2000) (1500 4000)
        RECT (1000 4500) (1500 6500)
        RECT (1000 7000) (1500 9000)
        RECT (1000 9500) (1500 11500) ;
END FILLS
```

The following `FILLS` statement defines two rectangles and one polygon fill geometries:

```
FILLS 1 ;
    -LAYER metal1
        RECT ( 100 200 ) ( 150 400 )
        POLYGON ( 100 100 ) ( 200 200 ) ( 300 200 ) ( 300 100 )
        RECT ( 300 200 ) ( 350 400 ) ;
END FILLS
```

GCell Grid

```
[GCELLGRID
     {X start DO numColumns+1 STEP space} ...
     {Y start DO numRows+1 STEP space ;} ...]
```

Defines the gcell grid for a standard cell-based design. Each `GCELLGRID` statement specifies a set of vertical (x) and horizontal (y) lines, or tracks, that define the gcell grid.

Typically, the `GCELLGRID` is automatically generated by a particular router, and is not manually created by the designer.

`DO` *`numColumns+1`*	Specifies the number of columns in the grid.
`DO` *`numRows+1`*	Specifies the number of rows in the grid.
`STEP` *`space`*	Specifies the spacing between tracks.
`X` *`start`*, `Y` *`start`*	Specify the location of the first vertical (x) and first horizontal (y) track.

GCell Grid Boundary Information

The boundary of the gcell grid is the rectangle formed by the extreme vertical and horizontal lines. The gcell grid partitions the routing portion of the design into rectangles, called gcells. The lower left corner of a gcell is the origin. The x size of a gcell is the distance between the upper and lower bounding vertical lines, and the y size is the distance between the upper and lower bounding horizontal lines.

For example, the grid formed by the following two `GCELLGRID` statements creates gcells that are all the same size (100 x 200 in the following):

```
    GCELLGRID X 1000 DO 101 STEP 100 ;
    GCELLGRID Y 1000 DO 101 STEP 200 ;
```

A gcell grid in which all gcells are the same size is called a uniform gcell grid. Adding `GCELLGRID` statements can increase the granularity of the grid, and can also result in a nonuniform grid, in which gcells have different sizes.

For example, adding the following two statements to the above grid generates a nonuniform grid:

```
    GCELLGRID X 3050 DO 61 STEP 100 ;
    GCELLGRID Y 5100 DO 61 STEP 200 ;
```

When a track segment is contained inside a gcell, the track segment belongs to that gcell. If a track segment is aligned on the boundary of a gcell, that segment belongs to the gcell only if it is aligned on the left or bottom edges of the gcell. Track segments aligned on the top or right edges of a gcell belong to the next gcell.

GCell Grid Restrictions

Every track segment must belong to a gcell, so gcell grids have the following restrictions:

- The x coordinate of the last vertical track must be less than, and not equal to, the x coordinate of the last vertical gcell line.
- The y coordinate of the last horizontal track must be less than, and not equal to, the y coordinate of the last horizontal gcell line.

Gcells grids also have the following restrictions:

- Each `GCELLGRID` statement must define two lines.
- Every gcell need not contain the vertex of a track grid. But, those that do must be at least as large in both directions as the default wire widths on all layers.

Groups

```
[GROUPS numGroups ;
     [– groupName compNameRegExpr ...
        [+ REGION regionNam]
        [+ PROPERTY {propName propVal} ...] ...
     ;] ...
END GROUPS]
```

Defines groups in a design.

`compNameRegExpr`	Specifies the components that make up the group. Do not assign any component to more than one group. You can specify any of the following: ■ A component name, for example `C3205` ■ A list of component names separated by spaces, for example, `I01 I02 C3204 C3205` ■ A regular expression for a set of components, for example, `IO*` and `C320*`
`groupName`	Specifies the name for a group of components.
`numGroups`	Specifies the number of groups defined in the `GROUPS` section.
`PROPERTY propName propVal`	Specifies a numerical or string value for a group property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.
`REGION regionName`	Specifies a rectangular region in which the group must lie. *`regionName`* specifies a region previously defined in the `REGIONS` section. If region restrictions are specified in both `COMPONENT` and `GROUP` statements for the same component, the component restriction overrides the group restriction.

History

```
[HISTORY anyText ;] ...
```

Lists a historical record about the design. Each line indicates one record. Any text excluding a semicolon (;) can be included in *`anyText`*. The semicolon terminates the `HISTORY` statement. `Linefeed` and `Return` do not terminate the `HISTORY` statement. Multiple `HISTORY` lines can appear in a file.

Nets

```
NETS numNets ;
     [– { netName
            [ ( {compName pinName | PIN pinName} [+ SYNTHESIZED] ) ] ...
        | MUSTJOIN ( compName pinName ) }
        [+ SHIELDNET shieldNetName ] ...
        [+ VPIN vpinName [LAYER layerName] pt pt
```

```
            [PLACED pt orient | FIXED pt orient | COVER pt orient] ] ...
        [+ SUBNET subnetName
            [ ( {compName pinName | PIN pinName | VPIN vpinName} ) ] ...
            [NONDEFAULTRULE rulename]
            [regularWiring] ...] ...
        [+ XTALK class]
        [+ NONDEFAULTRULE ruleName]
        [regularWiring] ...
        [+ SOURCE {DIST | NETLIST | TEST | TIMING | USER}]
        [+ FIXEDBUMP]
        [+ FREQUENCY frequency]
        [+ ORIGINAL netName]
        [+ USE {ANALOG | CLOCK | GROUND | POWER | RESET | SCAN | SIGNAL
                  | TIEOFF}]
        [+ PATTERN {BALANCED | STEINER | TRUNK | WIREDLOGIC}]
        [+ ESTCAP wireCapacitance]
        [+ WEIGHT weight]
        [+ PROPERTY {propName propVal} ...] ...
     ;] ...
END NETS
```

Defines netlist connectivity for nets containing regular pins. The default design rules apply to these pins, and the regular routers route to these pins. The `SPECIALNETS` statement defines netlist connectivity for nets containing special pins.

Input arguments for a net can appear in the `NETS` section or the `SPECIALNETS` section. In case of conflicting values, the DEF reader uses the last value encountered. `NETS` and `SPECIALNETS` statements can appear more than once in a DEF file. If a particular net has mixed wiring or pins, specify the special wiring and pins first.

compName pinName	Specifies the name of a regular component pin on a net or a subnet. LEF `MUSTJOIN` pins, if any, are not included; only the master pin (that is, the one without the `MUSTJOIN` statement) is included. If a subnet includes regular pins, the regular pins must be included in the parent net.
COVER *pt orient*	Specifies that the pin has a location and is a part of the cover macro. A `COVER` pin cannot be moved by automatic tools or by interactive commands. You must specify the pin's location and orientation.
ESTCAP *wireCapacitance*	Specifies the estimated wire capacitance for the net. `ESTCAP` can be loaded with simulation data to generate net constraints for timing-driven layout.

`FIXED` *`pt orient`*	Specifies that the pin has a location and cannot be moved by automatic tools, but can be moved by interactive commands. You must specify the pin's location and orientation.
`FIXEDBUMP`	Indicates that the bump net cannot be reassigned to a different pin. It is legal to have a pin without geometry to indicate a logical connection, and to have a net that connects that pin to two other instance pins that have geometry. Area I/Os have a logical pin that is connected to a bump and an input driver cell. The bump and driver cell have pin geometries (and, therefore, should be routed and extracted), but the logical pin is the external pin name without geometry (typically the Verilog pin name for the chip). Because bump nets are usually routed with special routing, they also can be specified in the `SPECIALNETS` statement. If a net name appears in both the `NETS` and `SPECIALNETS` statements, the `FIXEDBUMP` keyword also should appear in both statements. However, the value only exists once within a given application's database for the net name. Because DEF is often used incrementally, the last value read in is used. Therefore, in a typical DEF file, if the same net appears in both statements, the `FIXEDBUMP` keyword (or lack of it) in the `NETS` statement is the value that is used, because the `NETS` statement is defined after the `SPECIALNETS` statement. For an example specifying the `FIXEDBUMP` keyword, see "Fixed Bump" on page 182.
`FREQUENCY` *`frequency`*	Specifies the frequency of the net, in hertz. The frequency value is used by the router to choose the correct number of via cuts required for a given net, and by validation tools to verify that the AC current density rules are met. For example, a net described with `+ FREQUENCY 100` indicates the net has 100 rising and 100 falling transitions in 1 second. *Type:* Float
`LAYER` *`layerName`*	Specifies the layer on which the virtual pin lies.

`MUSTJOIN (`*`compName pinName`*`)`

Specifies that the net is a mustjoin. If a net is designated `MUSTJOIN`, its name is generated by the system. Only one net should connect to any set of mustjoin pins. Mustjoin pins for macros are defined in LEF. The only reason to specify a `MUSTJOIN` net in DEF (identified arbitrarily by one of its pins) is to specify prewiring for the `MUSTJOIN` connection.

Otherwise, nets are generated automatically where needed for mustjoin connections specified in the library. If the input file specifies that a mustjoin pin is connected to a net, the DEF reader connects the set of mustjoin pins to the same net. If the input file does not specify connections to any of the mustjoin pins, the DEF reader creates a local `MUSTJOIN` net.

`netName`

Specifies the name for the net. Each statement in the `NETS` section describes a single net. There are two ways of identifying the net: *`netName`* or `MUSTJOIN`. If the *`netName`* is given, a list of pins to connect to the net also can be specified. Each pin is identified by a component name and pin name pair (*`compName pinName`*) or as an I/O pin (`PIN` *`pinName`*). Parentheses ensure readability of output. The keyword `MUSTJOIN` cannot be used as a *`netName`*.

`NONDEFAULTRULE` *`ruleName`*

Specifies the LEF-defined nondefault rule to use when creating the net and wiring. When specified with `SUBNET`, identifies the nondefault rule to use when creating the subnet and its wiring.

`numNets`

Specifies the number of nets defined in the `NETS` section.

`ORIGINAL` *`netName`*

Specifies the original net partitioned to create multiple nets, including the net being defined.

`PATTERN {BALANCED | STEINER | TRUNK | WIREDLOGIC}`

Specifies the routing pattern used for the net.
Default: `STEINER`
Value: Specify one of the following:

`BALANCED`	Used to minimize skews in timing delays for clock nets.
`STEINER`	Used to minimize net length.
`TRUNK`	Used to minimize delay for global nets.

`WIREDLOGIC`	Used in ECL designs to connect output and mustjoin pins before routing to the remaining pins.

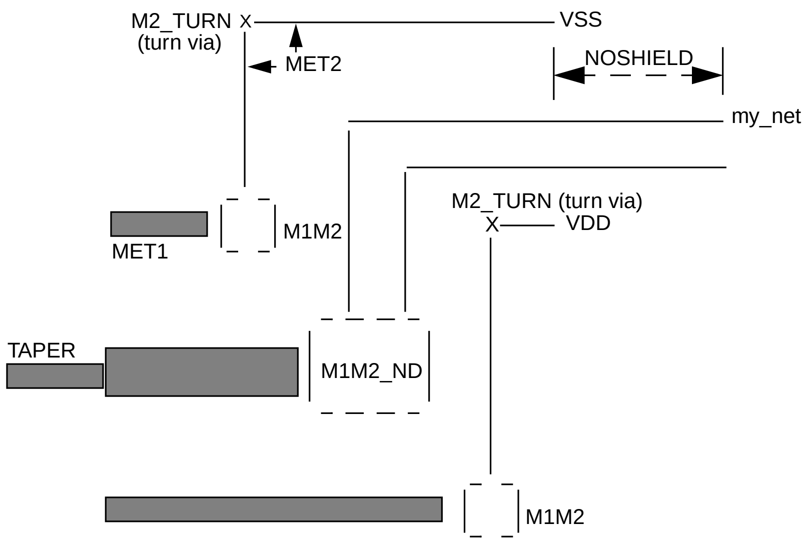
`PIN` *`pinName`*
Specifies the name of an I/O pin on a net or a subnet.

`PLACED` *`pt orient`*
Specifies that the pin has a location, but can be moved during automatic layout. You must specify the pin's location and orientation.

`PROPERTY` *`propName propVal`*
Specifies a numerical or string value for a net property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`regularWiring`
Specifies the regular physical wiring for the net or subnet. For regular wiring syntax, see "Regular Wiring Statement" on page 150.

`SHIELDNET` *`shieldNetName`*
Specifies the name of a special net that shields the regular net being defined. A shield net for a regular net is defined earlier in the DEF file in the `SPECIALNETS` section.



`SOURCE {DIST | NETLIST | TEST | TIMING | USER}`

Specifies the source of the net. The value of this field is preserved when input to the DEF reader.
Value: Specify one of the following:

`DIST`	Net is the result of adding physical components (that is, components that only connect to power or ground nets), such as filler cells, well-taps, tie-high and tie-low cells, and decoupling caps.
`NETLIST`	Net is defined in the original netlist. This is the default value, and is not normally written out in the DEF file.
`TEST`	Net is part of a scanchain.
`TIMING`	Net represents a logical rather than physical change to netlist, and is used typically as a buffer for a clock-tree, or to improve timing on long nets.
`USER`	Net is user defined.

`SUBNET` *`subnetName`*

Names and defines a subnet of the regular net *`netName`*. A subnet must have at least two pins. The subnet pins can be virtual pins, regular pins, or a combination of virtual and regular pins. A subnet pin cannot be a mustjoin pin.

`SYNTHESIZED`

Used by some tools to indicate that the pin is part of a synthesized scan chain.

`USE {ANALOG | CLOCK | GROUND | POWER | RESET | SCAN | SIGNAL | TIEOFF}`

Specifies how the net is used.
Value: Specify one of the following:

`ANALOG`	Used as an analog signal net.
`CLOCK`	Used as a clock net.
`GROUND`	Used as a ground net.
`POWER`	Used as a power net.
`RESET`	Used as a reset net.
`SCAN`	Used as a scan net.
`SIGNAL`	Used as a digital signal net.
`TIEOFF`	Used as a tie-high or tie-low net.

`VPIN` *`vpinName pt pt`*	Specifies the name of a virtual pin, and its physical geometry. Virtual pins can be used only in subnets. A `SUBNET` statement refers to virtual pins by the *`vpinName`* specified here. You must define each virtual pin in a `+ VPIN` statement before you can list it in a `SUBNET` statement.

Example 4-6   Virtual Pin

The following example defines a virtual pin:

```
+ VPIN M7K.v2 LAYER MET2 ( -10 -10 ) ( 10 10 ) FIXED ( 10 10 )
    + SUBNET M7K.2 ( VPIN M7K.v2 ) ( /PREG_CTRL/I$73/A I )
      NONDEFAULTRULE rule1
      ROUTED MET2 ( 27060 341440 ) ( 26880 * ) ( * 213280 )
      M1M2 ( 95040 * ) ( * 217600 ) ( 95280 * )
      NEW MET1 ( 1920 124960 ) ( 87840 * )
      COVER MET2 ( 27060 341440 ) ( 26880 * )
```

`WEIGHT` *`weight`*	Specifies the weight of the net. Automatic layout tools attempt to shorten the lengths of nets with high weights. A value of `0` indicates that the net length for that net can be ignored. The default value of `1` specifies that the net should be treated normally. A larger weight specifies that the tool should try harder to minimize the net length of that net. For normal use, timing constraints are generally a better method to use for controlling net length than net weights. For the best results, you should typically limit the maximum weight to 10, and not add weights to more than 3 percent of the nets. *Default:* `1` *Type:* Integer
`XTALK` *`class`*	Specifies the crosstalk class number for the net. If you specify the default value (`0`), `XTALK` will not be written to the DEF file. *Default:* `0` *Type:* Integer *Value:* `0` to `200`

Regular Wiring Statement

```
{+ COVER | + FIXED | + ROUTED | + NOSHIELD}
     layerName [TAPER | TAPERRULE ruleName] [STYLE styleNum]
        routingPoints
     [NEW layerName [TAPER | TAPERRULE ruleName] [STYLE styleNum]
        routingPoints
     ] ...
```

Specifies regular wiring for the net.

`COVER`	Specifies that the wiring cannot be moved by either automatic layout or interactive commands. If no wiring is specified for a particular net, the net is unrouted. If you specify `COVER`, you must also specify *`layerName`*.
`FIXED`	Specifies that the wiring cannot be moved by automatic layout, but can be changed by interactive commands. If no wiring is specified for a particular net, the net is unrouted. If you specify `FIXED`, you must also specify *`layerName`*.
`layerName`	Specifies the layer on which the wire lies. You must specify *`layerName`* if you specify `COVER`, `FIXED`, `ROUTED`, or `NEW`. Specified layers must be routable; reference to a cut layer generates an error.
`NEW` *`layerName`*	Indicates a new wire segment (that is, there is no wire segment between the last specified coordinate and the next coordinate), and specifies the name of the layer on which the new wire lies. Noncontinuous paths can be defined in this manner.
`NOSHIELD`	Specifies that the last wide segment of the net is not shielded. If the last segment is not shielded, and is tapered, specify `TAPER` under the `LAYER` argument, instead of `NOSHIELD`.
`ROUTED`	Specifies that the wiring can be moved by the automatic layout tools. If no wiring is specified for a particular net, the net is unrouted. If you specify `ROUTED`, you must also specify *`layerName`*.
`routingPoints`	Defines the center line coordinates of the route on *`layerName`*. For information about using routing points, see “Defining Routing Points” on page 152. The *`routingPoints`* syntax is defined as follows:

```
( x y [extValue] )
{ ( x y [extValue] ) | viaName [orient]} ...
```

`extValue`	Specifies the amount by which the wire is extended past the endpoint of the segment. The extension value must be greater than or equal to `0` (zero). *Default:* Half the wire width *Type:* Integer, specified in database units **Note:** Some tools only allow `0` or the `WIREEXTENSION` value from the `LAYER` or `NONDEFAULTRULE` statement.
`orient`	Specifies the orientation of the *`viaName`* that precedes it, using the standard DEF orientation values of `N`, `S`, `E`, `W`, `FN`, `FS`, `FE`, and `FW` (See “Specifying Orientation” on page 138). If you do not specify *`orient`*, `N` (North) is the default non-rotated value used. All other orientation values refer to the flipping or rotation around the via origin (the `0,0` point in the via shapes). The via origin is still placed at the `(`*`x y`*`)` value given in the routing statement just before the *`viaName`*. **Note:** Some tools do not support orientation of vias inside their internal data structures; therefore, they are likely to translate vias with an orientation into a different but equivalent via that does not require an orientation.
`viaName`	Specifies a via to place at the last point. If you specify a via, *`layerName`* for the next routing coordinates (if any) is implicity changed to the other routing layer for the via. For example, if the current layer is *metal1*, a *via12* changes the layer to *metal2* for the next routing coordinates.
`(` *`x y`* `)`	Specifies the route coordinates. For more information, see “Specifying Coordinates” on page 152. *Type:* Integer, specified in database units

`STYLE` *`styleNum`*	Specifies a previously defined style from the `STYLES` section in this DEF file. If a style is specified, the wire's shape is defined by the center line coordinates and the style.
`TAPER`	Specifies that the next contiguous wire segment on *`layerName`* is created using the default rule.
`TAPERRULE` *`ruleName`*	Specifies that the next contiguous wire segment on *`layerName`* is created using the specified nondefault rule.

Defining Routing Points

Routing points define the center line coordinates of the route for a specified layer. Routes that are 90 degrees, have a width defined by the routing rule for this wire, and extend from one coordinate (*`x y`*) to the next coordinate.

If either endpoint has an extension value (*`extValue`*), the wire is extended by that amount past the endpoint. Some applications require the extension value to be `0`, half of the wire width, or the same as the routing rule wire extension value. If you do not specify an extension value, the default value of half of the wire width is used.

If a coordinate with an extension value is specified after a via, the wire extension is added to the beginning of the next wire segment after the via (zero-length wires are not allowed).

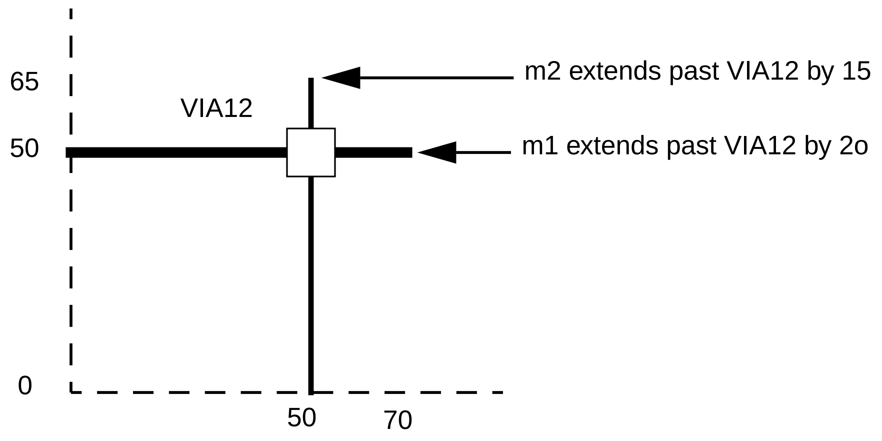
If the wire segment is a 45-degree edge, and no `STYLE` is specified, the default octagon style is used for the endpoints. The routing rule width must be an even multiple of the manufacturing grid in order to keep all of the coordinates of the resulting outer wire boundary on the manufacturing grid.

If a `STYLE` is defined for 90-degree or 45-degree routes, the routing shape is defined by the center line coordinates and the style. No corrections, such as snapping to manufacturing grid, can be applied, and any extension values are ignored. The DEF file should contain values that are already snapped, if appropriate. The routing rule width indicates the desired user width, and represents the minimum allowed width of the wire that results from the style when the 45-degree edges are properly snapped to the manufacturing grid.

Specifying Coordinates

To maximize compactness of the design files, the coordinates allow for the asterisk (`*`) convention. Here, (*`x`* `*`) indicates that the y coordinate last specified in the wiring specification is used; (`* y`) indicates that the x coordinate last specified is used. Use (`* * extValue`) to specify a wire extension at a via.

ROUTED M1 (0 50) (50 * 20) VIA12 (* * 15) (* 0)



Each coordinate sequence defines a connected orthogonal path through the points. The first coordinate in a sequence must not have an * element.

Because nonorthogonal segments are not allowed, subsequent points in a connected sequence must create orthogonal paths. For example, the following sequence is a valid path:

```
( 100 200 ) ( 200 200 ) ( 200 500 )
```

The following sequence is an equivalent path:

```
( 100 200 ) ( 200 * ) ( * 500 )
```

The following sequence is not valid because it represents a nonorthogonal segment.

```
( 100 200 ) ( 300 500 )
```

Specifying Orientation

If you specify the pin's placement status, you must specify its location and orientation. A pin can have any of the following orientations:

Value	Definition	Value	Definition
N	North	FN	Flipped north
S	South	FS	Flipped south
W	West	FW	Flipped west
E	East	FE	Flipped east

Example 4-7 Shielded Net

The following example defines a shielded net:

```
NETS 1 ;
    - my_net ( I1 CLK ) ( BUF OUT )
    + SHIELDNET VSS
    + SHIELDNET VDD
       ROUTED
    MET2 ( 14000 341440 ) ( 9600 * ) ( * 282400 )
    M1M2 ( 2400 * )
    + NOSHIELD MET2 ( 14100 341440 ) ( 14000 * )
    + TAPER MET1 ( 2400 282400 ) ( 240 * )
END NETS
```

Nondefault Rules

```
NONDEFAULTRULES numRules ;
     {- ruleName
           [+ HARDSPACING]
           {+ LAYER layerName
                WIDTH minWidth
                [DIAGWIDTH diagWidth]
                [SPACING minSpacing]
                [WIREEXT wireExt]
           } ...
           [+ VIA viaName] ...
           [+ VIARULE viaRuleName] ...
           [+ MINCUTS cutLayerName numCuts] ...
           [+ PROPERTY {propName propVal} ...] ...
     ;} ...
END NONDEFAULTRULES
```

Defines any nondefault rules used in this design that are not specified in the LEF file. This section can also contain the default rule and LEF nondefault rule definitions for reference. These nondefault rule names can be used anywhere in the DEF `NETS` section that requires a nondefault rule name.

If a nondefault rule name collides with an existing LEF or DEF nondefault rule name that has different parameters, the application should use the DEF definition when reading this DEF file, though it can change the DEF nondefault rule name to make it unique. This is typically done by adding a unique extension, such as `_1` or `_2` to the rule name.

All vias must be previously defined in the LEF `VIA` or DEF `VIAS` sections. Every nondefault rule must specify a width for every layer. If a nondefault rule does not specify a via or via rule for a particular routing-cut-routing layer combination, then there must be a `VIARULE GENERATE DEFAULT` rule that it inherited for that combination.

`DIAGWIDTH` *`diagWidth`*	Specifies the diagonal width for *`layerName`*, when 45-degree routing is used. *Default:* `0` (no diagonal routing allowed) *Type:* Integer, specified in DEF database units
`HARDSPACING`	Specifies that any spacing values that exceed the `LEF LAYER ROUTING` spacing requirements are "hard" rules instead of "soft" rules. By default, routers treat extra spacing requirements as soft rules that are high cost to violate, but not real spacing violations. However, in certain situations, the extra spacing should be treated as a hard, or real, spacing violation, such as when the route will be modified with a post-process that replaces some of the extra space with metal.
`LAYER` *`layerName`*	Specifies the layer for the various width and spacing values. *`layerName`* must be a routing layer. Each routing layer must have at least a minimum width specified.
`MINCUTS` *`cutLayerName numCuts`*	Specifies the minimum number of cuts allowed for any via using the specified cut layer. All vias (generated or fixed vias) used for this nondefault rule must have at least *`numCuts`* cuts in the via. *Type:* (*`numCuts`*) Positive integer
`numRules`	Specifies the number of nondefault rules defined in the `NONDEFAULTRULES` section.
`PROPERTY` *`propName propValue`*	Specifies a property for this nondefault rule. The *`propName`* must be defined as a `NONDEFAULTRULE` property in the `PROPERTYDEFINITIONS` section, and the *`propValue`* must match the type for *`propName`* (that is, integer, real, or string).
`rulename`	Specifies the name for this nondefault rule. This name can be used in the `NETS` section wherever a nondefault rule name is allowed. The reserved name `DEFAULT` can be used to indicate the default routing rule used in the `NETS` section.
`SPACING` *`minSpacing`*	Specifies the minimum spacing for *`layerName`*. The `LEF LAYER SPACING` or `SPACINGTABLE` definitions always apply;

	therefore it is only necessary to add a `SPACING` value if the desired spacing is larger than the `LAYER` rules already require. *Type:* Integer, specified in DEF database units.
`VIA` *`viaName`*	Specifies a previously defined LEF or DEF via to use with this rule.
`VIARULE` *`viaRuleName`*	Specifies a previously defined LEF `VIARULE GENERATE` to use with this routing rule. If no via or via rule is specified for a given routing-cut-routing layer combination, then a `VIARULE GENERATE DEFAULT` via rule must exist for that combination, and it is implicitly inherited.
`WIDTH` *`minWidth`*	Specifies the required minimum width allowed for *`layerName`*. *Type:* Integer, specified in DEF database units
`WIREEXT` *`wireExt`*	Specifies the distance by which wires are extended at vias on *`layerName`*. *Default:* `0` *Type:* Integer, specified in DEF database units

Example 4-8 Nondefault Rules

The following `NONDEFAULTRULES` statement is based on the assumption that there are `VIARULE GENERATE DEFAULT` rules for each routing-cut-routing combination, and that the default width is 0.3 µm.

```
NONDEFAULTRULES 5 ;
    - doubleSpaceRule #Needs extra space, inherits default via rules
        + LAYER metal1 WIDTH 0.2 SPACING 1.0
        + LAYER metal2 WIDTH 0.2 SPACING 1.0
        + LAYER metal3 WIDTH 0.2 SPACING 1.0 ;
    - lowerResistance #Wider wires and double cut vias for lower resistance
                      #and higher current capacity. No special spacing rules,
                      #therefore the normal LEF LAYER specified spacing rules
                      #apply. Inherits the default via rules.
        + LAYER metal1 WIDTH 0.6 #Metal1 is thinner, therefore a little wider
        + LAYER metal2 WIDTH 0.5
        + LAYER metal3 WIDTH 0.5
        + MINCUTS cut12 2 #Requires at least two cuts
        + MINCUTS cut23 2 ;
    - myRule             #Use default width and spacing, change via rules. The
                         #default via rules are not inherited.
        + LAYER metal1 WIDTH 0.2
        + LAYER metal2 WIDTH 0.2
        + LAYER metal3 WIDTH 0.2
        + VIARULE myvia12rule
        + VIARULE myvia23rule ;
    - myCustomRule       #Use new widths, spacing and fixed vias. The default
                         #via rules are not inherited because vias are defined.
        + LAYER metal1 WIDTH 0.5 SPACING 1.0
        + LAYER metal2 WIDTH 0.5 SPACING 1.0
        + LAYER metal3 WIDTH 0.5 SPACING 1.0
        + VIA myvia12_custom1
        + VIA myvia12_custom2
        + VIA myvia23_custom1
        + VIA myvia23_custom2 ;
END NONDEFAULTRULES
```

Pins

```
[PINS numPins ;
     [ [– pinName + NET netName]
         [+ SPECIAL]
         [+ DIRECTION {INPUT | OUTPUT | INOUT | FEEDTHRU}]
         [+ NETEXPR "netExprPropName defaultNetName"]
         [+ SUPPLYSENSITIVITY powerPinName]
         [+ GROUNDSENSITIVITY groundPinName]
         [+ USE {SIGNAL | POWER | GROUND | CLOCK | TIEOFF | ANALOG
                   | SCAN | RESET}]
         [+ ANTENNAPINPARTIALMETALAREA value [LAYER layerName]] ...
         [+ ANTENNAPINPARTIALMETALSIDEAREA value [LAYER layerName]] ...
         [+ ANTENNAPINPARTIALCUTAREA value [LAYER layerName]] ...
         [+ ANTENNAPINDIFFAREA value [LAYER layerName]] ...
         [+ ANTENNAMODEL {OXIDE1 | OXIDE2 | OXIDE3 | OXIDE4}] ...
         [+ ANTENNAPINGATEAREA value [LAYER layerName]] ...
         [+ ANTENNAPINMAXAREACAR value LAYER layerName] ...
         [+ ANTENNAPINMAXSIDEAREACAR value LAYER layerName] ...
         [+ ANTENNAPINMAXCUTCAR value LAYER layerName] ...
         [ + LAYER layerName
              [SPACING minSpacing | DESIGNRULEWIDTH effectiveWidth]
              pt pt
         | + POLYGON layerName
              [SPACING minSpacing | DESIGNRULEWIDTH effectiveWidth]
              pt pt pt ...] ...
         [+ COVER pt orient | FIXED pt orient | PLACED pt orient]
     ; ] ...
END PINS]
```

Defines external pins. Each pin definition assigns a pin name for the external pin and associates the pin name with a corresponding internal net name. The pin name and the net name can be the same.

When the design is a chip rather than a block, the `PINS` statement describes logical pins, without placement or physical information.

`ANTENNAMODEL {OXIDE1 | OXIDE2 | OXIDE3 | OXIDE4}`

Specifies the oxide model for the pin. If you specify an `ANTENNAMODEL` statement, that value affects all `ANTENNAGATEAREA` and `ANTENNA*CAR` statements for the pin that follow it until you specify another `ANTENNAMODEL` statement. The `ANTENNAMODEL` statement does not affect `ANTENNAPARTIAL*AREA` and `ANTENNADIFFAREA` statements because they refer to the total metal, cut, or diffusion area connected to the pin, and do not vary with each oxide model.
Default: `OXIDE1`, for a new `PIN` statement

Because DEF is often used incrementally, if an `ANTENNA` statement occurs twice for the same oxide model, the last value specified is used.

Usually, you only need to specify a few `ANTENNA` values; however, for a block with six routing layers, it is possible to have six different `ANTENNAPARTIAL*AREA` values and six different `ANTENNAPINDIFFAREA` values per pin. It is also possible to have six different `ANTENNAPINGATEAREA` and `ANTENNAPINMAX*CAR` values for each oxide model on each pin.

Example 4-9 Antenna Model Statement

The following example describes the `OXIDE1` and `OXIDE2` models for pin `clock1`. Note that the `ANTENNAPINPARTIALMETALAREA` and `ANTENNAPINDIFFAREA` values are not affected by the oxide values.

```
PINS 100 ;
   - clock1 + NET clock1
      ...
      + ANTENNAPINPARTIALMETALAREA 1.0 LAYER m1
      + ANTENNAPINDIFFAREA 0.5 LAYER m1
      ...
      + ANTENNAMODEL OXIDE1                      #not required, but good practice
      + ANTENNAPINGATEAREA 1.0
      + ANTENNAMAXAREACAR 300 LAYER m1
```

```
...
+ ANTENNAMODEL OXIDE2                     #start of OXIDE2 values
+ ANTENNAPINGATEAREA 2.0
+ ANTENNAMAXAREACAR 100 LAYER m1
...
```

`ANTENNAPINDIFFAREA` *`value`* `[LAYER` *`layerName`*`]`

Specifies the diffusion (diode) area to which the pin is connected on a layer. If you do not specify *`layerName`*, the value applies to all layers. This is not necessary for output pins.
Type: Integer
Value: Due to the maximum integer size, you can specify a value up to 2^{31}-1 in database units. In microns squared, this is 22^{31}-1 / DEFConvertFactor^{2}.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAPINGATEAREA` *`value`* `[LAYER` *`layerName`*`]`

Specifies the gate area to which the pin is connected on a layer. If you do not specify *`layerName`*, the value applies to all layers. This is not necessary for input pins.
Type: Integer
Value: Due to the maximum integer size, you can specify a value up to 2^{31}-1 in database units. In microns squared, this is 2^{31}-1 / DEFConvertFactor^{2}.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAPINMAXAREACAR` *`value`* `LAYER` *`layerName`*

For hierarchical process antenna effect calculation, specifies the maximum cumulative antenna ratio value, using the metal area at or below the current pin layer, excluding the pin area itself. Use this to calculate the actual cumulative antenna ratio on the pin layer, or the layer above it.
Type: Integer

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAPINMAXCUTCAR` *`value`* `LAYER` *`layerName`*

For hierarchical process antenna effect calculation, specifies the maximum cumulative antenna ratio value, using the cut area at or below the current pin layer, excluding the pin area itself. Use this to calculate the actual cumulative antenna ratio for the cuts above the pin layer.
Type: Integer

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAPINMAXSIDEAREACAR` *`value`* `LAYER` *`layerName`*

For hierarchical process antenna effect calculation, specifies the maximum cumulative antenna ratio value, using the metal side wall area at or below the current pin layer, excluding the pin area itself. Use this to calculate the actual cumulative antenna ratio on the pin layer, or the layer above it.
Type: Integer

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAPINPARTIALCUTAREA` *`value`* `[LAYER` *`cutLayerName`*`]`

Specifies the partial cut area above the current pin layer and inside the macro cell on a layer. If you do not specify *`layerName`*, the value applies to all layers. For hierarchical designs, only the cut layer above the I/O pin layer is needed for partial antenna ratio calculation.
Type: Integer
Value: Due to the maximum integer size, you can specify a value up to 2^{31}-1 in database units. In microns squared, this is 2^{31}-1 / DEFConvertFactor^{2}.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAPINPARTIALMETALAREA` *`value`* `[LAYER` *`layerName`*`]`

Specifies the partial metal area connected directly to the I/O pin and the inside of the macro cell on a layer. If you do not specify *`layerName`*, the value applies to all layers. For hierarchical designs, only the same metal layer as the I/O pin, or the layer above it, is needed for partial antenna ratio calculation.
Type: Integer
Value: Due to the maximum integer size, you can specify a value up to 2^{31}-1 in database units. In microns squared, this is 2^{31}-1 / DEFConvertFactor^2.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`ANTENNAPINPARTIALMETALSIDEAREA` *`value`* `[LAYER` *`layerName`*`]`

Specifies the partial metal side wall area connected directly to the I/O pin and the inside of the macro cell on a layer. If you do not specify *`layerName`*, the value applies to all layers. For hierarchical designs, only the same metal layer as the I/O pin, or the layer above it, is needed for partial antenna ratio calculation.
Type: Integer
Value: Due to the maximum integer size, you can specify a value up to 2^{31}-1 in database units. In microns squared, this is 2^{31}-1 / DEFConvertFactor^2.

For more information on process antenna calculation, see Appendix C, "Calculating and Fixing Process Antenna Violations,"

`COVER` *`pt orient`*

Specifies the pin's location, orientation, and that it is a part of the cover macro. A `COVER` pin cannot be moved by automatic tools or by interactive commands. If you specify a placement status for a pin, you must also include a `LAYER` statement.

`DESIGNRULEWIDTH` *`effectiveWidth`*

Specifies that this pin has a width of *`effectiveWidth`* for the purpose of spacing calculations. If you specify `DESIGNRULEWIDTH`, you cannot specify `SPACING`.
Type: Integer, specified in DEF database units

Example 4-10 Design Rule Width and Spacing Rules

The following statements create spacing rules using the DESIGNRULEWIDTH and SPACING statements:

```
PINS 3 ;
    - myPin1 + NET myNet1
        + DIRECTION INPUT
        + LAYER metal1
              DESIGNRULEWIDTH 1000   #Pin is effectively 1000 dbu wide
              ( -100 0 ) ( 100 200 ) #Pin is 200 x 200 dbu
        + FIXED ( 10000 5000 ) S ;
    - myPin2 + NET myNet2
        + DIRECTION INPUT
        + LAYER metal1
              SPACING 500            #Requires >= 500 dbu spacing
              ( -100 0 ) ( 100 200 ) #Pin is 200 x 200 dbu
        + COVER ( 10000 5000 ) S ;
    - myPin3 + NET myNet1            #Pin with two shapes
        + DIRECTION INPUT
        + LAYER metal2 ( 200 200 ) ( 300 300 ) #100 x 100 dbu shape
        + POLYGON metal1 ( 0 0 ) ( 100 100 ) ( 200 100 ) ( 200 0 ) #Has 45-degree edge
        + FIXED ( 10000 5000 ) N ;
END PINS
```

DIRECTION {INPUT | OUTPUT | INOUT | FEEDTHRU}

Specifies the pin type. Most current tools do not usually use this keyword. Typically, pin directions are defined by timing library data, and not from DEF.
Value: Specify one of the following:

INPUT	Pin that accepts signals coming into the cell.
OUTPUT	Pin that drives signals out of the cell.
INOUT	Pin that can accept signals going either in or out of the cell.
FEEDTHRU	Pin that goes completely across the cell.

FIXED *pt orient*

Specifies the pin's location, orientation, and that it's location cannot be moved by automatic tools, but can be moved by interactive commands. If you specify a placement status for a pin, you must also include a LAYER statement.

GROUNDSENSITIVITY *groundPinName*

Specifies that if this pin is connected to a tie-low connection (such as `1'b0` in Verilog), it should connect to the same net to which *`groundPinName`* is connected.

`groundPinName` must match another pin in this `PINS` section that has a `+ USE GROUND` attribute. The ground pin definition can follow later in this `PINS` section; it does not have to be defined before this pin definition. It is a semantic error to put this attribute on an existing ground pin. For an example, see Example 4-11 on page 164.

Note: `GROUNDSENSITIVITY` is useful only when there is more than one ground net connected to pins in the `PINS` section. By default, if there is only one net connected to all `+ USE GROUND` pins, the tie-low connections are already implicitly defined (that is, tie-low connections are connected to the same net as any ground pin).

`LAYER` *`layerName`* `...` *`pt pt`*

Specifies the routing layer used for the pin, and the pin geometry on that layer. If you specify a placement status for a pin, you must include a `LAYER` statement.

`NETEXPR "`*`netExprPropName defaultNetName`*`"`

Specifies a net expression property name (such as `power1` or `power2`) and a default net name. If *`netExprPropName`* matches a net expression property higher up in the netlist (for example, in Verilog, VHDL, or OpenAccess), then the property is evaluated, and the software identifies a net to which to connect this pin. If the property does not exist, *`defaultNetName`* is used for the net name.

`netExprPropName` must be a simple identifier in order to be compatible with other languages, such as Verilog and CDL. Therefore, it can only contain alphanumeric characters, and the first character cannot be a number. For example, `power2` is a legal name, but `2power` is not. You cannot use characters such as `$` and `!`. The *`defaultName`* can be any legal DEF net name.

If more than one pin connects to the same net, only one pin should have a `NETEXPR` added to it. It is redundant and unnecessary to add `NETEXPR` to every ground pin connected to one ground net, and it is illegal to have different `NETEXPR` values for pins connected to the same net.

Example 4-11 Net Expression and Supply Sensitivity

The following `PINS` statement defines sensitivity and net expression values for five pins in the design `myDesign`:

```
DESIGN myDesign
...
PINS 4 ;
    - in1 + NET myNet
        ...
        + SUPPLYSENSITIVITY vddpin1 ; #If in1 is connected to 1’b1, use
                                    #net that is connected to vddpin1.
                                    #No GROUNDSENSITIVITY is needed because
                                    #only one ground net is used by PINS.
                                    #Therefore, 1’b0 implicitly means net
                                    #from any +USE GROUND pin.
    - vddpin1 + NET VDD1 + USE POWER
        ...
        + NETEXPR "power1 VDD1" ; #If an expression named power1 is defined in
                                #the netlist, use it to fine the net.
                                #Otherwise, use net VDD1.
    - vddpin2 + NET VDD2 + USE POWER
        ...
        + NETEXPR "power2 VDD2" ; #If an expression named power2 is defined in
                                #the netlist, use it to find the net.
                                # Otherise, use net VDD2.
    - gndpin1 + NET GND + USE GROUND
        ...
        + NETEXPR "gnd1 GND" ; #If an expression named gnd1 is defined in
                             #the netlist, use it to find net
                             #connection. Otherwise, use net GND.
END PINS
```

`numPins` Specifies the number of pins defined in the `PINS` section.

`pinName` + `NET` *`netName`*

Specifies the name for the external pin, and the corresponding internal net (defined in `NETS` or `SPECIALNETS` statements).

`PLACED` *`pt orient`* Specifies the pin’s location, orientation, and that it’s location is fixed, but can be moved during automatic layout. If you specify a placement status for a pin, you must also include a `LAYER` statement.

`POLYGON` *`layerName ... pt pt pt`*

Specifies the layer and a sequence of at least three points to generate a polygon for this pin. The polygon edges must be parallel to the x axis, the y axis, or at a 45-degree angle. Each `POLYGON` statement defines a polygon generated by connecting each successive point, and then the first and last points. The *`pt`* syntax corresponds to a coordinate pair, such as *x y*. Specify an asterisk (`*`) to repeat the same value as the previous *x* or *y* value from the last point.

Example 4-12 Polygon

The following `PINS` statement creates a polygon with a 45-degree angle:

```
PINS 2 ;
    - myPin3 + NET myNet1
        + DIRECTION INPUT
        + POLYGON metal1 ( 0 0 ) ( 100 100 ) ( 200 100 ) ( 200 0 ) #45-degree angle
        + FIXED ( 10000 5000 ) N ;
    ...
END PINS
```

`SPACING` *`minSpacing`*

Specifies the minimum spacing allowed between this pin and any other routing shape. This distance must be greater than or equal to *`minSpacing`*. If you specify `SPACING`, you cannot specify `DESIGNRULEWIDTH`. For an example using the `SPACING` keyword, see Example 4-10 on page 161.
Type: Integer, specified in DEF database units

`SPECIAL`

Identifies the pin as a special pin. Regular routers do not route to special pins. The special router routes special wiring to special pins.

`SUPPLYSENSITIVITY` *`powerPinName`*

Specifies that if this pin is connected to a tie-high connection (such as `1'b1` in Verilog), it should connect to the same net to which *`powerPinName`* is connected.

`powerPinName` must match another pin in this `PINS` section that has a `+ USE POWER` attribute. The power pin definition can follow later in this `PINS` section; it does not have to be defined before this pin definition. It is a semantic error to put this attribute on an existing power pin. For an example, see Example 4-11 on page 164.

Note: `POWERSENSITIVITY` is useful only when there is more than one power net connected to pins in the `PINS` section. By default, if there is only one net connected to all `+ USE POWER` pins, the tie-high connections are already implicitly defined (that is, tie-high connections are connected to the same net as the single power pin).

`USE {ANALOG | CLOCK | GROUND | POWER | RESET | SCAN | SIGNAL | TIEOFF}`

Specifies how the pin is used.
Default: `SIGNAL`
Value: Specify one of the following:

`ANALOG`	Pin is used for analog connectivity.
`CLOCK`	Pin is used for clock net connectivity.
`GROUND`	Pin is used for connectivity to the chip-level ground distribution network.
`POWER`	Pin is used for connectivity to the chip-level power distribution network.
`RESET`	Pin is used as reset pin.
`SCAN`	Pin is used as scan pin.
`SIGNAL`	Pin is used for regular net connectivity.
`TIEOFF`	Pin is used as tie-high or tie-low pin.

Extra Physical PIN(S) for One Logical PIN

In the design of place and route blocks, you sometimes want to add extra physical connection points to existing signal ports (usually to enable the signal to be accessed from two sides of the block). One pin has the same name as the net it is connected to. This pin is related to the pins specified in GCF for boundary conditions. Any other pins added to the net must use the following naming conventions.

For extra non-bus bit pin names, use the following syntax:

`pinname``.extra`*`N`*	*`N`* is a positive integer, incremented as the physical pins are added

For example:

```
    PINS n ;
        - a + NET a .... ;
        - a.extra1 + NET a ... ;
```

For extra bus bit pin names, use the following syntax:

`basename``.extra`*`N`*`[`*`index`*`]`

`basename` is simple part of bus bit pin/net name*.* *`N`* is a positive integer, incremented as the physical pins are added. [*`index`*] identifies the specific bit of the bus

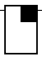
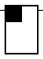
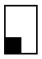
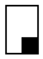
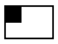
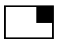
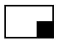
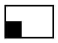
For example:

```
    PINS n ;
        - a[0] + net a[0] ... ;
        - a.extra1[0] + net a[0] ... ;
```

Note: The brackets [ ] are the `BUSBITCHARS` as defined in the DEF `BUSBITCHARS` statement.

Specifying Orientation

If you specify the pin's placement status, you must specify its location and orientation. A pin can have any of the following orientations:

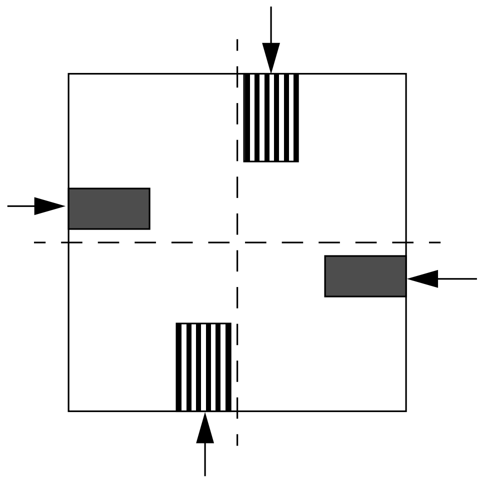
Value	Definition	Value	Definition
N	North 	FN	Flipped north 
S	South 	FS	Flipped south 
W	West 	FW	Flipped west 
E	East 	FE	Flipped east 

Example 4-13 Pin Statements

The following example describes a physical I/O pin.

```
# M1 width = 50, track spacing = 120
# M2 width = 60, track spacing = 140
```

```
DIEAREA ( -5000 -5000 ) ( 5000 5000 ) ;
TRACKS Y -4900 DO 72 STEP 140 LAYER M2 M1 ;
TRACKS X -4900 DO 84 STEP 120 LAYER M1 M2 ;
PINS 4 ;
    # Pin on the left side of the block
    - BUSA[0]+ NET BUSA[0] + DIRECTION INPUT
                + LAYER M1 ( -25 0 ) ( 25 165 ) # .5 M1 W + 1 M2 TRACK
                + PLACED ( -5000 2500 ) E ;
    # Pin on the right side of the block
    - BUSA[1] + NET BUSA[1] + DIRECTION INPUT
                + LAYER M1 ( -25 0 ) ( 25 165 ) # .5 M1 W + 1 M2 TRACK
                + PLACED ( 5000 -2500 ) W ;
    # Pin on the bottom side of the block
    - BUSB[0] + NET BUSB[0] + DIRECTION INPUT
                + LAYER M2 ( -30 0 ) ( 30 150 ) # .5 M2 W + 1 M1 TRACK
                + PLACED ( -2100 -5000 ) N ;
    # Pin on the top side of the block
    - BUSB[1] + NET BUSB[1] + DIRECTION INPUT
                + LAYER M2 ( -30 0 ) ( 30 150 ) # .5 M2 W + 1 M1 TRACK
                + PLACED ( 2100 5000 ) S ;
END PINS
```



The following example shows how a logical I/O pin would appear in the DEF file. The pin is first defined in Verilog for a chip-level design.

```
module chip (OUT, BUSA, BUSB) ;
    input [0:1] BUSA, BUSB;
    output OUT;
....
```

```
endmodule
```

The following description for this pin is in the `PINS` section in the DEF file:

```
PINS 5 ;
    - BUSA[0] + NET BUSA[0] + DIRECTION INPUT ;
    - BUSA[1] + NET BUSA[1] + DIRECTION INPUT ;
    - BUSB[0] + NET BUSB[0] + DIRECTION INPUT ;
    - BUSB[1] + NET BUSB[1] + DIRECTION INPUT ;
    - OUT + NET OUT + DIRECTION OUTPUT ;
END PINS
```

Pin Properties

```
[PINPROPERTIES num;
     [- {compName pinName | PIN pinName}
           [+ PROPERTY {propName propVal} ...] ...
     ;] ...
END PINPROPERTIES]
```

Defines pin properties in the design.

compName pinName	Specifies a component pin. Component pins are identified by the component name and pin name.
num	Specifies the number of pins defined in the `PINPROPERTIES` section.
`PIN` *pinName*	Specifies an I/O pin.
`PROPERTY` *propName propVal*	Specifies a numerical or string value for a pin property defined in the `PROPERTYDEFINITIONS` statement. The *propName* you specify must match the *propName* listed in the `PROPERTYDEFINITIONS` statement.

Example 4-14 Pin Properties Statement

```
PINPROPERTIES 3 ;
    - CORE/g76 CKA + PROPERTY CLOCK "FALLING" ;
    - comp1 A + PROPERTY CLOCK "EXCLUDED" ;
    - rp/regB clk + PROPERTY CLOCK "INSERTION" ;
END PINPROPERTIES
```

Property Definitions

```
[PROPERTYDEFINITIONS
     [objectType propName propType [RANGE min max]
        [value | stringValue]
     ;] ...
END PROPERTYDEFINITIONS]
```

Lists all properties used in the design. You must define properties in the `PROPERTYDEFINITIONS` statement before you can refer to them in other sections of the DEF file.

`objectType`	Specifies the object type being defined. You can define properties for the following object types:
	`COMPONENT`
	`COMPONENTPIN`
	`DESIGN`
	`GROUP`
	`NET`
	`NONDEFAULTRULE`
	`REGION`
	`ROW`
	`SPECIALNET`

`propName`	Specifies a unique property name for the object type.

`propType`	Specifies the property type for the object type. You can specify one of the following property types:
	`INTEGER`
	`REAL`
	`STRING`

`RANGE` *`min max`*	Limits real number and integer property values to a specified range.
`value | stringValue`	Assigns a numeric value or a name to a `DESIGN` object.

Note: Assign values to properties for component pins in the `PINPROPERTIES` section. Assign values to other properties in the section of the LEF file that describes the object to which the property applies.

Regions

```
[REGIONS numRegions ;
     [- regionName {pt pt} ...
        [+ TYPE {FENCE | GUIDE}]
        [+ PROPERTY {propName propVal} ...] ...
     ;] ...
END REGIONS]
```

Defines regions in the design. A region is a physical area to which you can assign a component or group.

`numRegions` Specifies the number of regions defined in the design.

`PROPERTY` *`propName propVal`*
Specifies a numerical or string value for a region property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`regionName pt pt` Names and defines a region. You define a region as one or more rectangular areas specified by pairs of coordinate points.

`TYPE {FENCE | GUIDE}`
Specifies the type of region.
Default: All instances assigned to the region are placed inside the region boundaries, and other cells are also placed inside the region.
Value: Specify one of the following:

`FENCE`	All instances assigned to this type of region must be exclusively placed inside the region boundaries. No other instances are allowed inside this region.

`GUIDE`	All instances assigned to this type of region should be placed inside this region; however, it is a preference, not a hard constraint. Other constraints, such as wire length and timing, can override this preference.

Example 4-15 Regions Statement

```
REGIONS 1 ;
    - REGION1 ( 0 0 ) ( 1200 1200 )
        + PROPERTY REGIONORDER 1 ;
```

Rows

```
[ROW rowName siteName origX origY siteOrient
       [DO numX BY numY [STEP stepX stepY]]
       [+ PROPERTY {propName propVal} ...] ... ;] ...
```

Defines rows in the design.

`DO` *`numX`* `BY` *`numY`*	Specifies a repeating set of sites that create the row. You must specify one of the values as `1`. If you specify `1` for *`numY`*, then the row is horizontal. If you specify `1` for *`numX`*, the row is vertical. *Default:* Both *`numX`* and *`numY`* equal `1`, creating a single site at this location (that is, a horizontal row with one site).
`origX origY`	Specifies the location of the first site in the row. Type: Integer, specified in DEF database units
`PROPERTY` *`propName propVal`*	Specifies a numerical or string value for a row property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.
`rowName`	Specifies the row name for this row.
`siteName`	Specifies the LEF `SITE` to use for the row. A site is a placement location that can be used by LEF macros that match the same site. *`siteName`* can also refer to a site with a row pattern in its definition, in which case, the row pattern indicates a repeating set of sites that are abutted. For more information, see "Site" and "Macro" in "LEF Syntax."

`siteOrient`	Specifies the orientation of all sites in the row. *`siteOrient`* must be one of `N`, `S`, `E`, `W`, `FN`, `FS`, `FE`, or `FW`. For more information on orientations, see "Specifying Orientation" on page 138.
`STEP` *`stepX stepY`*	Specifies the spacing between sites in horizontal and vertical rows.

Example 4-16 Row Statements

Assume `siteA` is 200 by 900 database units.

```
ROW row_0 siteA 1000 1000 N ; #Horizontal row is one-site wide at 1000, 1000
ROW row_1 siteA 1000 1000 N DO 1 BY 1 ;            #Same as row_0
ROW row_2 siteA 1000 1000 N DO 1 BY 1 STEP 200 0 ; #Same as row_0
ROW row_3 siteA 1000 1000 N DO 10 BY 1 ; #Horizontal row is 10 sites wide,
                                         #so row width is 200*10=2000 dbu
ROW row_4 siteA 1000 1000 N DO 10 BY 1 STEP 200 0 ; #Same as row_3
ROW row_5 siteA 1000 1000 N DO 1 BY 10 ; #Vertical row is 10 sites high, so
                                         #total row height is 900*10=9000 dbu
ROW row_6 siteA 1000 1000 N DO 1 BY 10 STEP 0 900 ; #Same as row_5
```

Scan Chains

```
[SCANCHAINS numScanChains ;
     [- chainName
        [+ PARTITION partitionName [MAXBITS maxbits]]
        [+ COMMONSCANPINS [ ( IN pin )] [( OUT pin ) ] ]
         + START {fixedInComp | PIN} [outPin]
        [+ FLOATING
           {floatingComp [ ( IN pin ) ] [ ( OUT pin ) ] [ ( BITS numBits ) ]} ...]
        [+ ORDERED
           {fixedComp [ ( IN pin ) ] [ ( OUT pin ) ] [ ( BITS numBits ) ]} ...
        ] ...
         + STOP {fixedOutComp | PIN} [inPin] ]
     ;] ...
END SCANCHAINS]
```

Defines scan chains in the design. Scan chains are a collection of cells that contain both scan-in and scan-out pins. These pins must be defined in the `PINS` section of the DEF file with + `USE SCAN`.

`chainName`	Specifies the name of the scan chain. Each statement in the `SCANCHAINS` section describes a single scan chain.

`COMMONSCANPINS [( IN` *`pin`* `)] [( OUT` *`pin`* `)]`

Specifies the scan-in and scan-out pins for each component that does not have a scan-in and scan-out pin specified. You must specify either common scan-in and scan-out pins, or individual scan-in and scan-out pins for each component.

`FLOATING {`*`floatingComp`* `[( IN` *`pin`* `)] [( OUT` *`pin`* `)] [( BITS` *`numBits`* `)]}`

Specifies the floating list. You can have one or zero floating lists. If you specify a floating list, it must contain at least one component.

`fixedComp`	Specifies the component name.
`( IN` *`pin`* `)`	Specifies the scan-in pin. If you do not specify a scan-in pin, the router uses the pin you specified for the common scan pins.
`( OUT` *`pin`* `)`	Specifies the scan-out pin. If you do not specify a scan-out pin, the router uses the pin you specified for the common scan pins.
`BITS` *`numBits`*	Specifies the sequential bit length of any chain element. This allows application tools that do not have library access to determine the sequential bit length contribution of any chain element to ensure the `MAXBITS` constraints are not violated for chains in a given partition. You can specify `0` to indicate when elements are nonsequential. *Default:* `1` *Type:* Integer

Note: Scan chain reordering commands can use floating components in any order to synthesize a scan chain. Floating components cannot be shared with other scan chains unless they are in the same `PARTITION`. Each component should only be used once in synthesizing a scan chain.

`MAXBITS` *`maxBits`*

When specified with chains that include the `PARTITION` keyword, sets the maximum bit length (flip-flop bit count) that the chain can grow to in the partition.
Default: `0` (tool-specific defaults apply, which is probably the

highest bit length of any chain in the partition
Type: Integer
Value: Specify a value that is at least as large as the size of the current chain.

`numScanChains` Specifies the number of scan chains to synthesize.

`ORDERED {fixedComp [( IN pin )] [( OUT pin )] [( BITS numBits )]}`

Specifies an ordered list. You can specify none or several ordered lists. If you specify an ordered list, you must specify at least two fixed components for each ordered list.

`fixedComp`	Specifies the component name.
`( IN pin )`	Specifies the scan-in pin. If you do not specify a scan-in pin, the router uses the pin you specified for the common scan pins.
`( OUT pin )`	Specifies the scan-out pin. If you do not specify a scan-out pin, the router uses the pin you specified for the common scan pins.
`BITS numBits`	Specifies the sequential bit length of any chain element. This allows application tools that do not have library access to determine the sequential bit length contribution of any chain element to ensure the `MAXBITS` constraints are not violated for chains in a given partition. You can specify `0` to indicate when elements are nonsequential. *Default:* `1` *Type:* Integer

Note: Scan chain reordering commands should synthesize these components in the same order that you specify them in the list. Ordered components cannot be shared with other scan chains unless they are in the same `PARTITION`. Each component should only be used once in synthesizing a scan chain.

`PARTITION partitionName`

Specifies a partition name. This statement allows reordering tools to determine inter-chain compatibility for element swapping (both `FLOATING` elements and `ORDERED` elements). It associates each chain with a partition group, which determines their compatibility for repartitioning by swapping elements between them.

Chains with matching `PARTITION` names constitute a swap-compatible group. You can change the length of chains included in the same partition (up to the `MAXBITS` constraint on the chain), but you cannot eliminate chains or add new ones; the number of chains in the partition is always preserved.

If you do not specify the `PARTITION` keyword, chains are assumed to be in their own single partition, and reordering can be performed only within that chain.

Example 4-17 Partition Scanchain

In the following definition, chain `chain1_clock1` is specified without a `MAXBITS` keyword. The maximum allowed bit length of the chain is assumed to be the sequential length of the longest chain in any `clock1` partition.

```
SCANCHAINS 77 ;
    - chain1_clock1
        + PARTITION clock1
        + START block1/bsr_reg_0 Q
        + FLOATING
           block1/pgm_cgm_en_reg_reg ( IN SD ) ( OUT QZ )
           ...
           block1/start_reset_dd_reg ( IN SD ) ( OUT QZ )
        + STOP block1/start_reset_d_reg SD ;
```

In the following definition, chain `chain2_clock2` is specified with a `PARTITION` statement that associates it with `clock2`, and a maximum bit length of `1000`. The third element statement in the `FLOATING` list is a scannable register bank that has a sequential bit length of `4`. The `ORDERED` list element statements have total bit lengths of `1` each because the muxes are specified with a maximum bit length of `0`.

```
    - chain2_clock2
        + PARTITION clock2
          MAXBITS 1000
        + START block1/current_state_reg_0_QZ
        + FLOATING
           block1/port2_phy_addr_reg_0_ ( IN SD ) ( OUT QZ )
           block1/port2_phy_addr_reg_4_ ( IN SD ) ( OUT QZ )
           block1/port3_intfc ( IN SD ) ( OUT MSB ) ( BITS 4 )
                ...
        + ORDERED
           block1/mux1 ( IN A ) ( OUT X ) ( BITS 0 )
           block1/ff1 ( IN SD ) ( OUT Q )
        + ORDERED
           block1/mux2 ( IN A ) ( OUT X ) ( BITS 0 )
           block1/ff2 ( IN SD ) ( OUT Q ) ;
```

In the following definition, chain `chain3_clock2` is also specified with a `PARTITION` statement that associates it with `clock2`. This means it is swap-compatible with `chain2_clock2`. The specified maximum bit length for this chain is `1200`.

```
    - chain3_clock2
        + PARTITION clock2
          MAXBITS 1200
        + START block1/LV_testpoint_0_Q_reg Q
        + FLOATING
           block1/LV_testpoint_0_Q_reg  ( IN SE ) ( OUT Q )
           block1/tm_state_reg_1_ (IN SD ) ( OUT QZ )
           ...
```

In the following definition, chain `chain4_clock3` is specified with a `PARTITION` statement that associates it with `clock3`. The second element statement in the `FLOATING` list is a scannable register bank that has a sequential bit length of 8, and default pins. The `ORDERED` list element statements have total bit lengths of 2 each because the mux is specified with a maximum bit length of `0`.

```
    - chain4_clock3
        + PARTITION clock3
        + START block1/prescaler_IO/lfsr_reg1
        + FLOATING
           block1/dp1_timers
           block1/bus8 ( BITS 8 )
           ...
        + ORDERED
           block1/ds1/ff1 ( IN SD ) ( OUT Q )
           block1/ds1/mux1 ( IN B ) ( OUT Y ) ( BITS 0 )
           block1/ds1/ff2 ( IN SD ) ( OUT Q )
           ...
```

`START {`*`fixedInComp`*` | PIN} [`*`outPin`*`]`

Specifies the start point of the scan chain. You must specify this point. The starting point can be either a component, *`fixedInComp`*, or an I/O pin, `PIN`. If you do not specify *`outPin`*, the router uses the pin specified for common scan pins.

`STOP {`*`fixedOutComp`*` | PIN} [`*`inPin`*`]`

Specifies the endpoint of the scan chain. You must specify this point. The stop point can be either a component, *`fixedOutComp`*, or an I/O pin, `PIN`. If you do not specify *`inPin`*, the router uses the pin specified for common scan pins.

Scan Chain Rules

Note the following when defining scan chains.

- Each scan-in/scan-out pin pair of adjacent components in the ordered list cannot have different owning nets.
- No net can connect a scan-out pin of one component to the scan-in pin of a component in a different scan chain.
- For incremental DEF, if you have a `COMPONENTS` section and a `SCANCHAINS` section in the same DEF file, the `COMPONENTS` section must appear before the `SCANCHAINS` section. If the `COMPONENTS` section and `SCANCHAINS` section are in different DEF files, you must read the `COMPONENTS` section or load the database before reading the `SCANCHAINS` section.

Example 4-18 Scan Chain Statements

```
Nets 100; #Number of nets resulting after scan chain synthesis
    - SCAN-1 ( C1 SO + SYNTHESIZED )
             ( C4 SI + SYNTHESIZED ) + SOURCE TEST ;
    - ...
    - N1 ( C3 SO + SYNTHESIZED )
         ( C11 SI + SYNTHESIZED ) ( AND1 A ) ;
    - ...
END NETS
SCANCHAINS 2; #Specified before scan chain ordering
    - S1
        + COMMONSCANPINS ( IN SI ) ( OUT SO )
        + START SIPAD OUT
        + FLOATING C1 C2 ( IN D ) ( OUT Q ) C3 C4 C5...CN
        + ORDERED A1 ( OUT Q ) A2 ( IN D ) ( OUT Q ) ...
               AM ( N D )
        + ORDERED B1 B2 ... BL
        + STOP SOPAD IN ;
    - S2 ... ;
END SCANCHAINS
SCANCHAINS 2 ; #Specified after scan chain ordering
    - S1
        + START SIPAD OUT
        + FLOATING C1 ( IN SI ) ( OUT SO )
            C2 ( IN D ) ( OUT Q )
            C3 ( IN SI ( OUT SO ) ... CN ( IN SI ) ( OUT SO )
        + ORDERED A1 ( IN SI ) ( OUT Q )
            A2 ( IN D ) ( OUT Q ) ... AM ( IN D ) ( OUT SO )
        + ORDERED B1 ( IN SI ) ( OUT SO )
            B2 ( IN SI ) ( OUT SO ) ...
        + STOP SOPAD IN ;
    - S2 ... ;
END SCANCHAINS
```

Slots

```
[SLOTS numSlots ;
     [- LAYER layerName
         {RECT pt pt | POLYGON pt pt pt ... } ...
     ;] ...
END SLOTS]
```

Defines the rectangular shapes that form the slotting of the wires in the design. Each slot is defined as an individual rectangle.

LAYER *layerName*	Specifies the layer on which to create slots.
numSlots	Specifies the number of LAYER statements in the SLOTS statement, *not* the number of rectangles.
POLYGON *pt pt pt*	Specifies a sequence of at least three points to generate a polygon geometry. The polygon edges must be parallel to the x axis, the y axis, or at a 45-degree angle. Each POLYGON statement defines a polygon generated by connecting each successive point, and then the first and last points. The *pt* syntax corresponds to a coordinate pair, such as *x y*. Specify an

	asterisk (`*`) to repeat the same value as the previous *x* or *y* value from the last point. *Type:* DEF database units
`RECT` *`pt pt`*	Specifies the lower left and upper right corner coordinates of the slot geometry.

Example 4-19 Slots Statements

The following statement defines slots for layers `MET1` and `MET2`.

```
SLOTS 2 ;
    - LAYER MET1
        RECT (1000 2000) (1500 4000)
        RECT (2000 2000) (2500 4000)
        RECT (3000 2000) (3500 4000) ;
    - LAYER MET2
        RECT (1000 2000) (1500 4000)
        RECT (1000 4500) (1500 6500)
        RECT (1000 7000) (1500 9000)
        RECT (1000 9500) (1500 11500) ;
END SLOTS
```

The following `SLOTS` statement defines two rectangles and one polygon slot geometries:

```
SLOTS 1 ;
    - LAYER metal1
        RECT ( 100 200 ) ( 150 400 )
        POLYGON ( 100 100 ) ( 200 200 ) ( 300 200 ) ( 300 100 )
        RECT ( 300 200 ) ( 350 400 ) ;
END SLOTS
```

Special Nets

```
[SPECIALNETS numNets ;
     [– netName [ ( compNameRegExpr pinName [+ SYNTHESIZED] ) ] ...
       [+ VOLTAGE volts]
       [specialWiring] ...
       [+ SOURCE {DIST | NETLIST | TIMING | USER}]
       [+ FIXEDBUMP]
       [+ ORIGINAL netName]
       [+ USE {ANALOG | CLOCK | GROUND | POWER | RESET | SCAN | SIGNAL | TIEOFF}]
       [+ PATTERN {BALANCED | STEINER | TRUNK | WIREDLOGIC}]
       [+ ESTCAP wireCapacitance]
       [+ WEIGHT weight]
       [+ PROPERTY {propName propVal} ...] ...
     ;] ...
END SPECIALNETS]
```

Defines netlist connectivity for nets containing special pins. Each specification in the `SPECIALNETS` statement describes a single net, identified by *netName* and the special pins on the net. These pins are identified by their pin names and corresponding components.

Input parameters for a net can appear in the `NETS` section or the `SPECIALNETS` section. In case of conflicting values for an argument, the DEF reader uses the last value encountered for the argument. `NETS` and `SPECIALNETS` statements can appear more than once in a DEF file. If a particular net has mixed wiring or pins, specify the special wiring and pins first.

You can also specify the netlist in the `COMPONENTS` statement. If the netlist is specified in both `NETS` and `COMPONENTS` statements, and if the specifications are not consistent, an error message appears. On output, the writer outputs the netlist in either format, depending on the command arguments of the output command.

compNameRegExpr pinName
: Specifies the name of a special pin on the net and its corresponding component. You can use a regular expression to specify a set of component names. During evaluation of the regular expression, components that match the expression but do not have a pin named *pinName* are ignored.

`ESTCAP` *wireCapacitance*
: Specifies the estimated wire capacitance for the net. `ESTCAP` can be loaded with simulation data to generate net constraints for timing-driven layout.

`FIXEDBUMP`
: Indicates that the bump net cannot be reassigned to a different pin.

It is legal to have a pin without geometry to indicate a logical connection and to have a net that connects that pin to two other instance pins that have geometry. Area I/Os have a logical pin that is connected to a bump and an input driver cell. The bump and driver cell have pin geometries (and, therefore, should be routed and extracted), but the logical pin is the external pin name without geometry (typically the Verilog pin name for the chip).

Bump nets also can be specified in the `NETS` statement. If a net name appears in both the `NETS` and `SPECIALNETS` statements, the FIXEDBUMP keyword also should appear in both statements. However, the value only exists once within a given application's database for the net name.

Because DEF is often used incrementally, the last value read in is used. Therefore, in a typical DEF file, if the same net appears in both statements, the FIXEDBUMP keyword (or lack of it) in the NETS statement is the value that is used because the NETS statement is defined after the SPECIALNETS statement.

Example 4-20 Fixed Bump

The following example describes a logical pin that is connected to a bump and an input driver cell. The I/O driver cell and bump cells are specified in the COMPONENTS statement. Bump cells are usually placed with + COVER placement status so they cannot be moved manually by mistake.

```
COMPONENTS 200
    - driver1 drivercell + PLACED ( 100 100 ) N ;
    ...
    - bumpa1 bumpcell + COVER ( 100 100 ) N ;
    - bumpa2 bumpcell + COVER ( 200 100 ) N ;
```

The pin is assigned in the PIN statement.

```
PINS 100
    - n1 + NET n1 + SPECIAL + DIRECTION INPUT ;
    - n2 + NET n2 + SPECIAL + DIRECTION INPUT ;
```

In the SPECIALNETS statement, the net n1 is assigned to bumpa1 and cannot be reassigned. Note that another net n2 is assigned to bumpa2; however, I/O optimization commands are allowed to reassign bumpa2 to a different net.

```
SPECIALNETS 100
    - n1 ( driver1 in ) ( bumpa1 bumpin ) + FIXEDBUMP ;
    - n2 ( driver2 in ) ( bumpa2 bumpin ) ;
```

netName	Specifies the name of the net.
ORIGINAL *netName*	Specifies the original net partitioned to create multiple nets, including the current net.
PATTERN {BALANCED | STEINER | TRUNK | WIREDLOGIC}	Specifies the routing pattern used for the net.

Default: `STEINER`
Value: Specify one of the following:

`BALANCED`	Used to minimize skews in timing delays for clock nets.
`STEINER`	Used to minimize net length.
`TRUNK`	Used to minimize delay for global nets.
`WIREDLOGIC`	Used in ECL designs to connect output and mustjoin pins before routing to the remaining pins.

`PROPERTY` ***`propName propVal`***

Specifies a numerical or string value for a net property defined in the `PROPERTYDEFINITIONS` statement. The *`propName`* you specify must match the *`propName`* listed in the `PROPERTYDEFINITIONS` statement.

`specialWiring` Specifies the special wiring for the net. For syntax information, see “Special Wiring Statement” on page 184.

`SOURCE {DIST | NETLIST | TIMING | USER}`

Specifies how the net is created. The value of this field is preserved when input to the DEF reader.

`DIST`	Net is the result of adding physical components (that is, components that only connect to power or ground nets), such as filler cells, well-taps, tie-high and tie-low cells, and decoupling caps.
`NETLIST`	Net is defined in the original netlist. This is the default value, and is not normally written out in the DEF file.
`TEST`	Net is part of a scanchain.
`TIMING`	Net represents a logical rather than physical change to netlist, and is used typically as a buffer for a clock-tree, or to improve timing on long nets.
`USER`	Net is user defined.

`SYNTHESIZED` Used by some tools to indicate that the pin is part of a synthesized scan chain.

`USE {ANALOG | CLOCK | GROUND | POWER | RESET | SCAN | SIGNAL | TIEOFF}`

Specifies how the net is used.
Value: Specify one of the following:

`ANALOG`	Used as an analog signal net.
`CLOCK`	Used as a clock net.
`GROUND`	Used as a ground net.
`POWER`	Used as a power net.
`RESET`	Used as a reset net.
`SCAN`	Used as a scan net.
`SIGNAL`	Used as a digital signal net.
`TIEOFF`	Used as a tie-high or tie-low net.

`VOLTAGE` *`volts`*
Specifies the voltage for the net, as an integer in units of .001 volts. For example, `VOLTAGE 1500` in DEF is equal to 1.5 V.

`WEIGHT` *`weight`*
Specifies the weight of the net. Automatic layout tools attempt to shorten the lengths of nets with high weights. Do not specify a net weight larger than 10, or assign weights to more than 3 percent of the nets in a design.

Note: The net constraints method of controlling net length is preferred over using net weights.

Special Wiring Statement

```
 [    + POLYGON layerName pt pt pt ...
    | + RECT layerName pt pt
    | {+ COVER | + FIXED | + ROUTED | + SHIELD shieldNetName}
           layerName  routeWidth
              [+ SHAPE {RING | PADRING | BLOCKRING | STRIPE | FOLLOWPIN
                  | IOWIRE | COREWIRE | BLOCKWIRE | BLOCKAGEWIRE | FILLWIRE
                  | DRCFILL}]
              [+ STYLE styleNum]
              routingPoints
           [NEW layerName routeWidth
              [+ SHAPE {RING | PADRING | BLOCKRING | STRIPE | FOLLOWPIN
                  | IOWIRE | COREWIRE | BLOCKWIRE | BLOCKAGEWIRE | FILLWIRE
                  | DRCFILL}]
              [+ STYLE styleName]
              routingPoints
           ] ...
```

```
  ] ...
```

Defines the wiring for both routed and shielded nets.

`COVER`

Specifies that the wiring cannot be moved by either automatic layout or interactive commands. If no wiring is specified for a particular net, the net is unrouted. If you specify `COVER`, you must also specify *`layerName width`*.

`FIXED`

Specifies that the wiring cannot be moved by automatic layout, but can be changed by interactive commands. If no wiring is specified for a particular net, the net is unrouted. If you specify `FIXED`, you must also specify *`layerName width`*.

`layerName routeWidth`

Specifies the width for wires on *`layerName`*. Specified layers must be routable; reference to a cut layer generates an error. For more information, see “Defining Routing Points” on page 189.

Vias do not change the route width. When a via is used in special wiring, the previously established *`routeWidth`* is used for the next wire in the new layer. To change the *`routeWidth`*, a new path must be specified using `NEW` *`layerName routeWidth`*.

Many applications require *`routeWidth`* to be an even multiple of the manufacturing grid in order to be fabricated, and to keep the center line on the manufacturing grid.
Type: Integer, specified in database units

`NEW` *`layerName routewidth`*

Indicates a new wire segment (that is, that there is no wire segment between the last specified coordinate and the next coordinate) on *`layerName`*, and specifies the width for the wire. Noncontinuous paths can be defined in this manner. For more information, see “Defining Routing Points” on page 189.
Type: Integer, specified in database units

`POLYGON` *`layerName pt pt pt`*

Specifies a sequence of at least three points to generate a polygon geometry on *`layerName`*. The polygon edges must be parallel to the x axis, the y axis, or at a 45-degree angle. Each polygon statement defines a polygon generated by connecting each successive point, then connecting the first and last points. The *`pt`* syntax corresponds to a coordinate pair, such as *`x y`*.

	Specify an asterisk (`*`) to repeat the same value as the previous *x* or *y* value from the last point. *Type:* (*x y*) Integer, specified in database units
`RECT` *`layerName pt pt`*	Specifies a rectangle on layer *`layerName`*. The two points define opposite corners of the rectangle. The *`pt`* syntax corresponds to a coordinate pair, such as *x y*. You cannot define the same *x* and *y* values for both points (that is, a zero-area rectangle is not legal). *Type:* (*x y*) Integer, specified in database units
`ROUTED`	Specifies that the wiring can be moved by automatic layout tools. If no wiring is specified for a particular net, the net is unrouted. If you specify `ROUTED`, you must also specify *`layerName width`*.
`routingPoints`	Defines the center line coordinates of the route on *`layerName`*. For information on and an example of using routing points, see "Defining Routing Points" on page 189, and Example 4-22 on page 188.

The *`routingPoints`* syntax is defined as follows:

```
( x y [extValue])
 {( x y [extValue])
  | viaName
     [DO numX BY numY STEP stepX stepY]
 } ...
 DO numx BY numY STEP stepX stepY
```

	Creates an array of power vias of the via specified with `viaName`. *`numX`* and *`numY`* specify the number of vias to create, in the x and y directions. Do not specify `0` as a value. *Type:* Integer *`stepX`* and *`stepY`* specify the step distance between vias, in the x and y directions, in DEF distance database units. *Type:* Integer For an example of a via array, see Example 4-21 on page 187.
`extValue`	Specifies the amount by which the wire is extended past the endpoint of the segment. *Type:* Integer, specified in database units
`viaName`	Specifies a via to place at the last point. If you specify a via, *`layerName`* for the next routing coordinates (if any) is implicity changed to the other routing layer for the via. For example, if the current layer is *metal1*, a *via12* changes the layer to *metal2* for the next routing coordinates.
`(` *`x y`* `)`	Specifies the route coordinates. For more information, see "Specifying Coordinates" on page 190. *Type:* Integer, specified in database units

Example 4-21 Via Arrays

The following example specifies arrays of via `VIAGEN21_2` on *metal1* and *metal2*.

```
SPECIALNETS 2 ;
    -vdd ( * vdd )
       + ROUTED metal1 150 ( 100 100 ) ( 200 * )
       NEW metal1 0 ( 200 100 ) VIAGEN21_2 DO 10 BY 20 STEP 10000 20000
       NEW metal2 0 (-900 -30 ) VIAGEN21_2 DO 1000 BY 1 STEP 5000 0
       ...
```

As with any other `VIA` statement, the `DO` statement does not change the previous coordinate. Therefore, the following statement creates a *metal1* wire of width 50 from (200 100) to (200 200) along with the via array that starts at (200 100).

```
    NEW metal1 50 ( 200 100 ) VIAGEN21_2 DO 10 BY 20 STEP 1000 2000 ( 200 200 )
```

Example 4-22 Special Wiring

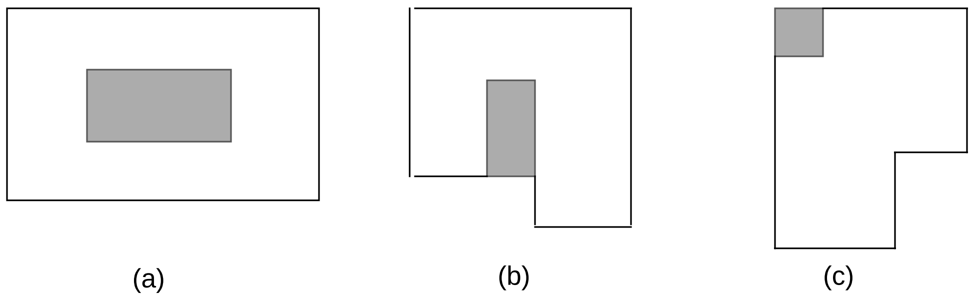
```
SPECIALNETS 1 ;
    - vdd (*vdd)
        + USE POWER
        + POLYGON metal1 ( 0 0 ) ( 0 100 ) ( 100 100 ) ( 200 200 ) ( 200 0 )
        + POLYGON metal2 ( 100 100 ) ( * 200 ) ( 200 * ) ( 300 300 ) ( 300 100 )
        + RECT metal1 ( 0 0 ) ( 100 200 )
        + ROUTED metal1 100 ( 0 0 50 ) ( 100 0 50 ) via12 ( 100 100 50 )
        + ROUTED metal2 100 + SHAPE RING + STYLE 1 ( 0 0 ) ( 100 100 ) ( 200 100 )
        ;
END SPECIALNETS
```

`SHAPE`

Specifies a wire with special connection requirements because of its shape. This applies to vias as well as wires.
Value: Specify one of the following:

`RING`	Used as ring, target for connection
`PADRING`	Connects padrings
`BLOCKRING`	Connects rings around the blocks
`STRIPE`	Used as stripe
`FOLLOWPIN`	Connects standard cells to power structures.
`IOWIRE`	Connects I/O to target
`COREWIRE`	Connects endpoints of followpin to target
`BLOCKWIRE`	Connects block pin to target
`BLOCKAGEWIRE`	Connects blockages
`FILLWIRE`	Connects metal fills to power and ground.
	Represents fill wire that is connected to special net
`DRCFILL`	Used as a fill shape to correct DRC errors, such as `SPACING`, `MINENCLOSEDAREA`, or `MINSTEP` violations on wires and pins of the same net (see Figure 4-2 on page 189.)

Figure 4-2 Fill Shapes



Examples of fill inside (a) a `MINENCLOSEDAREA` violation, (b) a `SPACING` violation, and (c) a `MINSTEP` violation.

`SHIELD` *`shieldNetName`*

Specifies the name of a regular net to be shielded by the special net being defined.

After describing shielded routing for a net, use `+ ROUTED` to return to the routing of the special net being defined.

`STYLE` *`styleNum`*
Specifies a previously defined style number from the `STYLES` section in this DEF file. The style is used with the endpoints of each routing segment to define the routing shape, and applies to all routing segments defined in one *`routingPoints`* statement.

Defining Routing Points

Routing points define the center line coordinates of a route. If a route has a 90-degree edge, it has a width of *`routeWidth`*, and extends from one coordinate (*`x y`*) to the next coordinate.

If either endpoint has an optional extension value (*`extValue`*), the wire is extended by that amount past the endpoint. If a coordinate with an extension value is specified after a via, the wire extension is added to the beginning of the next wire segment after the via (zero-length wires are not allowed). Some applications convert the extension value to an equivalent route that has the *`x`* and *`y`* points already extended, with no extension value.

The *`routeWidth`* must be an even value to ensure that the corners of the route fall on a legal database coordinate without round off. Because most vendors specify a manufacturing grid, *routeWidth* must be an even multiple of the manufacturing grid in order to be fabricated.

If the wire segment is a 45-degree edge, and no STYLE is specified, the default octagon style is used for the endpoints. The *routeWidth* must be an even multiple of the manufacturing grid in order to keep all of the coordinates of the resulting outer wire boundary on the manufacturing grid.

If a STYLE is defined for 90-degree or 45-degree routes, the routing shape is defined by the center line coordinates and the style. No corrections, such as snapping to manufacturing grid, should be applied, and any extension values are ignored. The DEF file should contain values that are already snapped, if appropriate. The *routeWidth* indicates the desired user width, and represents the minimum allowed width of the wire that results from the style when the 45-degree edges are snapped to the manufacturing grid. See Figure 4-4 on page 195 through Figure 4-12 on page 204 for examples.

Specifying Coordinates

To maximize compactness of the design files, the coordinates allow for the asterisk (*) convention. For example, (*x* *) indicates that the y coordinate last specified in the wiring specification is used; (* *y*) indicates that the x coordinate last specified is used.

Each coordinate sequence defines a connected orthogonal or 45-degree path through the points. The first coordinate in a sequence must not have an * element.

All subsequent points in a connected sequence must create orthogonal or 45-degree paths. For example, the following sequence is a valid path:

```
( 100 200 ) ( 200 200 ) ( 200 500 )
```

The following sequence is an equivalent path:

```
( 100 200 ) ( 200 * ) ( * 500 )
```

The following sequence is not valid because it is not an orthogonal or 45-degree segment.

```
( 100 200 ) ( 300 500 )
```

Special Pins and Wiring

Pins that appear in the SPECIALNETS statement are special pins. Regular routers do not route to these pins. The special router routes special wiring to special pins. If you use a component-based format to input the connectivity for the design, special pins to be routed by the special router also must be specified in the SPECIALNETS statement, because pins included in the COMPONENTS statement are considered regular.

The following example inputs connectivity in a component-based format, specifies VDD and VSS pins as special pins, and marks VDD and VSS nets for special routing:

```
COMPONENTS 3 ;
  C1 AND N1 N2 N3 ;
  C2 AND N4 N5 N6 ;
END COMPONENTS

SPECIALNETS 2 ;
  VDD ( * VDD ) + WIDTH M1 5 ;
  VSS ( * VSS ) ;
END SPECIALNETS
```

Shielded Routing

If, in a non-routed design, a net has + SHIELDNET attributes, the router adds shielded routing to this net. + NOSHIELD indicates the last wide segment of the net is not shielded. If the last segment is not shielded and is tapered, use the + TAPER keyword instead of + NOSHIELD. For example:

```
+ SHIELDNET VSS      # both sides will be shielded with VSS
+ SHIELDNET VDD      # one side will be shielded with VDD and
+ SHIELDNET VSS     # one side will be shielded with VSS
```

After you add shielded routing to a special net, it has the following syntax:

```
+ SHIELD regularNetName
  MET2 regularWidth ( x y )
```

A shield net specified for a regular net should be defined earlier in the DEF file in the SPECIALNETS section. After describing shielded routing for a net, use + ROUTED to return to the routing of the current special net.

For example:

```
SPECIALNETS 2 ;
    - VSS
       + ROUTED MET2 200
    ...
    + SHIELD my_net MET2 100 ( 14100 342440 ) ( 13920 * )
          M2_TURN ( * 263200 ) M1M2 ( 2400 * ) ;
    - VDD
       + ROUTED MET2 200
    ...
    + SHIELD my_net MET2 100 ( 14100 340440 ) ( 8160 * )
```

```
         M2_TURN ( * 301600 ) M1M2 ( 2400 * );
END SPECIALNETS
```

Styles

```
[STYLES numStyles ;
     {- STYLE styleNum pt pt pt ... ;} ...
END STYLES]
```

Defines a convex polygon that is used at each of the endpoints of a wire to precisely define the wire's outer boundary. A style polygon consists of three to eight points. Informally, half of the style polygon defines the first endpoint wire boundary, and the other half of the style polygon defines the second endpoint wire boundary. Octagons and squares are the most common styles.

numStyles Specifies the number of styles specified in the STYLES section.

STYLE *styleNum pt pt pt*

Defines a new style. *styleNum* is an integer that is greater than or equal to 0 (zero), and is used to reference the style later in the DEF file. When defining multiple styles, the first *styleNum* must be 0 (zero), and any following *styleNum* should be numbered consecutively so that a table lookup can be used to find them easily.

Style numbers are keys used locally in the DEF file to reference a particular style, but not actual numbers preserved in the application. Each style number must be unique. Style numbers can only be used inside the same DEF file, and are not preserved for use in other DEF files. Because applications are not required to preserve the style number itself, an application that writes out an equivalent DEF file might use different style numbers.
Type: Integer

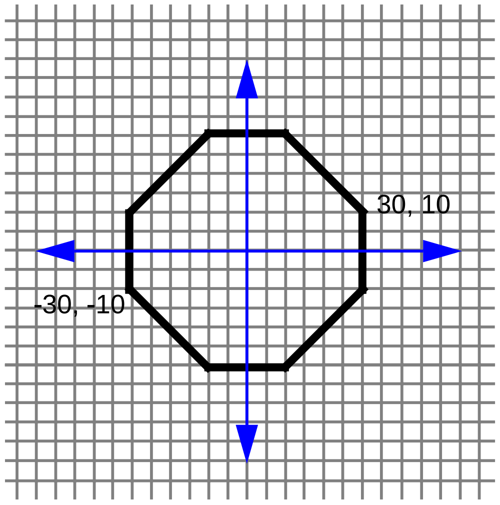
The *pt* syntax specifies a sequence of at least three points to generate a polygon geometry. The syntax corresponds to a coordinate pair, such as *x y*. Specify an asterisk (`*`) to repeat the same value as the previous *x* (or *y*) value from the last point. The polygon must be convex. The polygon edges must be parallel to the x axis, the y axis, or at a 45-degree angle, and must enclose the point (`0 0`).
Type: Integer, specified in DEF database units

Example 4-23 Styles Statement

The following STYLES statement defines the basic octagon shown in Figure 4-3 on page 193.

```
STYLES 1 ;
    - STYLE 1 ( 30 10 ) ( 10 30 ) ( -10 30 ) ( -30 10 ) ( -30 -10 )
                ( -10 -30 ) ( 10 -30 ) ( 30 -10 ) ;
END STYLES
```

Figure 4-3



Defining Styles

A style is defined as a polygon with points P1 through P*n*. The center line is given as (X0, Y0) to (X1, Y1). Two sets of points are built (P0,1 through P0,*n* and P1,1 through P1,*n*) as follows:

```
    P0,i = Pi + (X0, Y0) for 1 <= i <= n
    P1,i = Pi + (X1, Y1) for 1 <= i <= n
```

The resulting wire segment shape is a counterclockwise, eight-sided polygon (S1 through S8) that can be computed in the following way:

S1 = lowest point in (left-most points in (P0,1 through P0,*n* P1,1 through P1,*n*))

S2 = left-most point in (lowest points in (P0,1 through P0,*n* P1,1 through P1,*n*))

S3 = right-most point in (lowest points in (P0,1 through P0,*n* P1,1 through P1,*n*))

S4 = lowest point in (right-most points in (P0,1 through P0,*n* P1,1 through P1,*n*))

S5 = highest point in (right-most points in (P0,1 through P0,*n* P1,1 through P1,*n*))

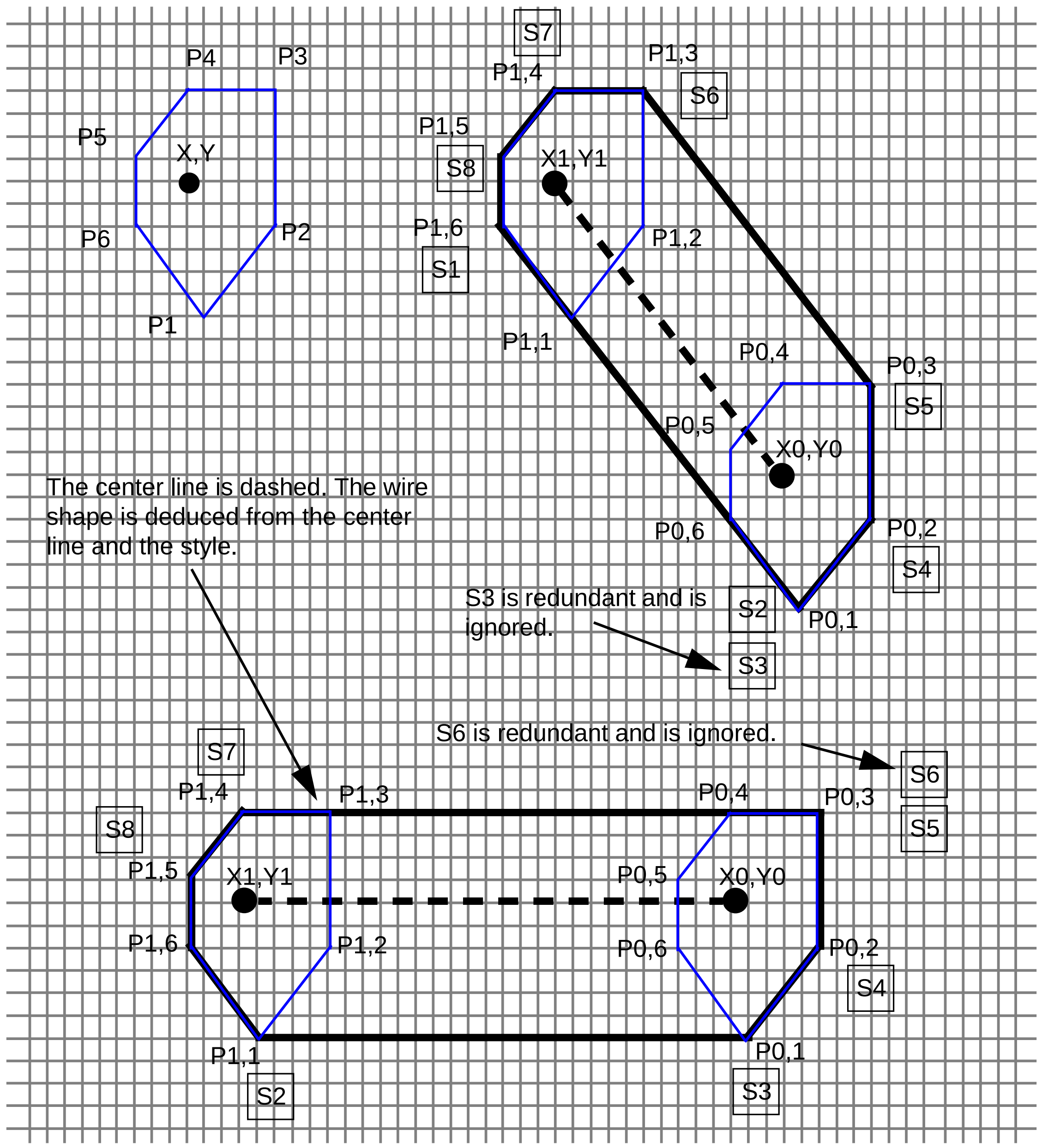
S6 = right-most point in (highest points in (P0,1 through P0,*n* P1,1 through P1,*n*))

S7 = left-most point in (highest points in (P0,1 through P0,*n* P1,1 through P1,*n*))

S8 = highest point in (left-most points in (P0,1 through P0,*n* P1,1 through P1,*n*))

When consecutive points are collinear, only one of them is relevant, and the resulting shape has less than eight sides, as shown in Figure 4-4 on page 195. A more advanced algorithm can order the points and only have to check a subset of the points, depending on which endpoint was used, and whether the wire was horizontal, vertical, a 45-degree route, or a 135-degree route.

Figure 4-4



Examples of X Routing with Styles

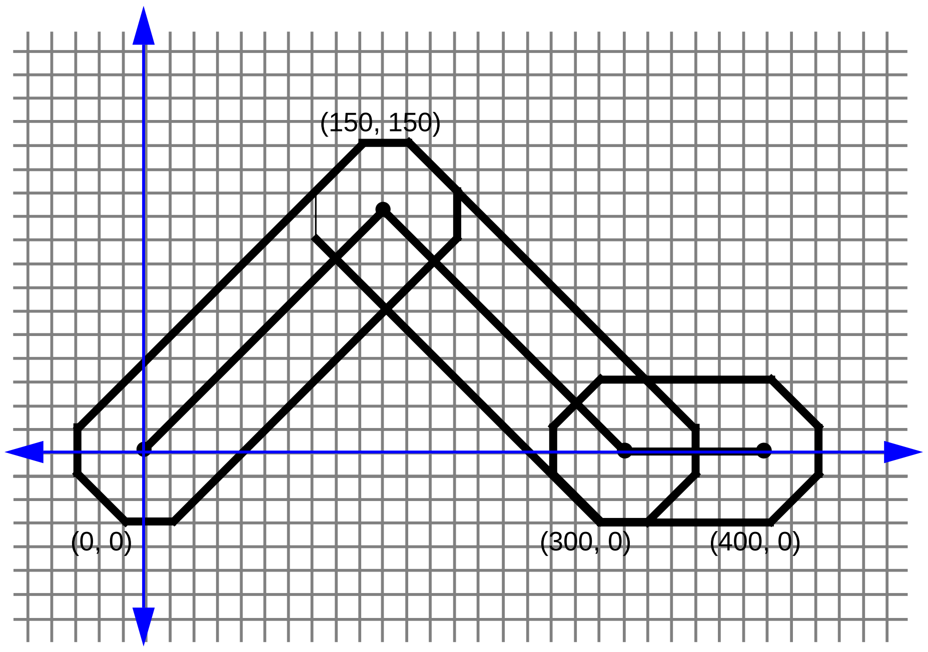
The following examples illustrate the use of styles for X routing. In two cases, there are examples of `SPECIALNETS` syntax and `NETS` syntax that result in the same geometry.

Example 1

The following statements define an X wire with octagonal ends, as shown in Figure 4-5 on page 196.

```
STYLES 1 ;
    - STYLE 0 ( 30 10 ) ( 10 30 ) ( -10 30 ) ( -30 10 ) ( -30 -10 ) ( -10 -30 )
         ( 10 -30 ) ( 30 -10 ) ;          #An octagon.
END STYLES

SPECIALNETS 1 ;
    - VSS ...
        + ROUTED metal3 50 + STYLE 0 ( 0 0 ) ( 150 150 ) ( 300 0 ) ( 400 0 ) ;
          #The style applies to all the segments until a NEW statement or ";"
          #at the end of the net.
END SPECIALNETS

NETS 1 ;
    - mySignal ...
        + ROUTED metal3 STYLE 0 ( 0 0 ) ( 150 150 ) ( 300 0 ) ( 400 0 ) ;
        #The style applies to all the segments in the ROUTED statement
END NETS
```

Figure 4-5

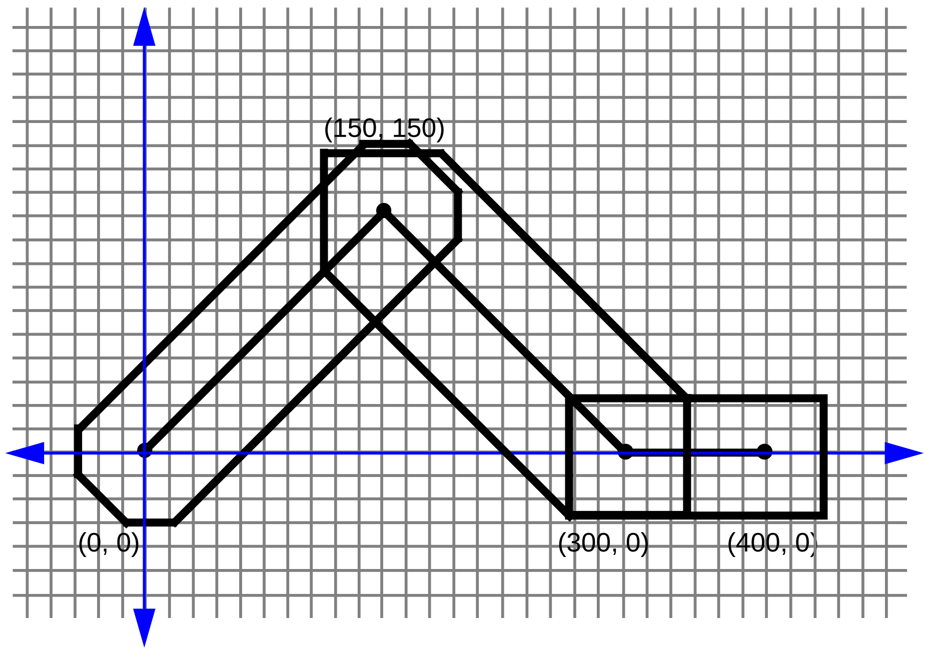


Example 2

The following statements define the same X wire with mixed octagonal and manhattan styles, as shown in Figure 4-6 on page 197.

```
STYLES 2 ;
    - STYLE 0 ( 30 10 ) ( 10 30 ) ( -10 30 ) ( -30 10 ) ( -30 -10 ) ( -10 -30 )
        ( 10 -30 ) ( 30 -10 ) ;                                 #An octagon
    - STYLE 1 ( 25 25 ) ( -25 25 ) ( -25 -25 ) ( 25 -25 ) ;  #A square
END STYLES

SPECIALNETS 1 ;
    - POWER (* power)
        + ROUTED metal3 50 + STYLE 0 ( 0 0 ) ( 150 150 )
        NEW metal3 50 + STYLE 1 ( 150 150 ) ( 300 0 ) ( 400 0 ) ;
END SPECIALNETS

NETS 1 ;
    - mySignal ...
        + ROUTED metal3 STYLE 0 ( 0 0 ) ( 150 150 )
        NEW metal3 STYLE 1 ( 150 150 ) ( 300 0 ) ( 400 0 ) ;
END NETS
```

Figure 4-6



Note: The square ends might be necessary for connecting to manhattan wires or pins, or in cases where vias have a manhattan shape even on X routing layers. In practice, the middle wire probably would not use a simple square, such as `style2`; it would use a combination of an octagon and a square for the middle segment style, in order to smooth out the resulting outline at the (150,150) point.

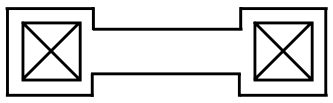
Example 3

The following statements define a manhattan wire with a width of 70, as shown in Figure 4-7 on page 198.

This example emphasizes that the style overrides the width of 100 units. In this case, the style polygon is a square 70 x 70 units wide, and the vias (`via12`) are 100 x 100 units wide. The application that creates the styles is responsible for meeting any particular width requirements. Normally, the resulting style-computed width is equal to or larger than the wire width given in the routing statement.

```
STYLES 1 ;
    - STYLE 0 ( 35 35 ) ( -35 35 ) ( -35 -35 ) ( 35 -35 ) ;
END STYLES

SPECIALNETS 1 ;
    - POWER ...
        + ROUTED metal1 100 + STYLE 0 ( 0 0 ) via12 ( 600 * ) via12 ;
END SPECIALNETS
```

Figure 4-7

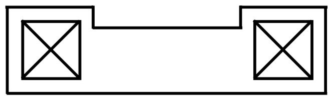


Example 4

The following statements define a similar wire that is offset from the center, as shown in Figure 4-8 on page 199. Similar to Example 3, the center line in both runs through the middle of the X in the vias.

```
STYLES 1 ;
    - STYLE 0 ( 35 20 ) ( -35 20 ) ( -35 -50 ) ( 35 -50 ) ; #70 x 70 offset square
END STYLES

SPECIALNETS 1 ;
    - POWER ...
        + ROUTED metal1 100 + STYLE 0 ( 0 0 ) via12 ( 600 * ) via12 ;
END SPECIALNETS
```

Figure 4-8

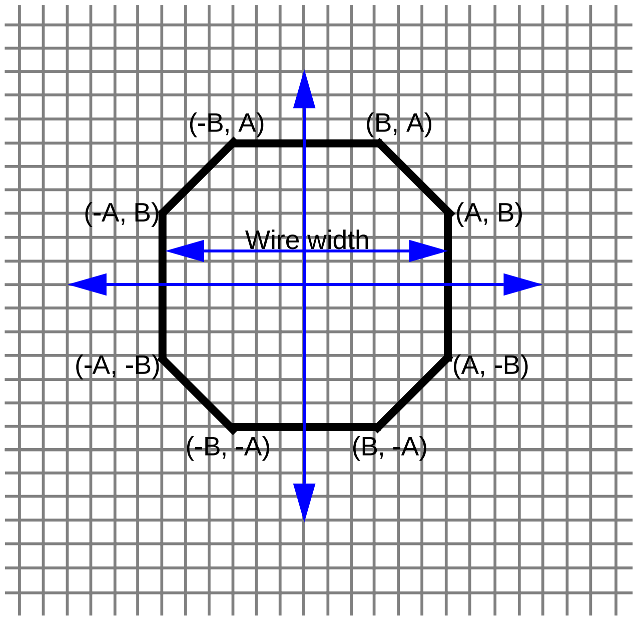


45-Degree Routing Without Styles

Because many applications only store the wire endpoints and the width of the wire, DEF includes a specific style default definition. If a style is not explicitly defined, the default style is implicitly included with any 45-degree routing segment. It is computed directly from the wire width and endpoints, at the expense of some loss in flexibility.

The default style is an octagon (shown in Figure 4-9 on page 199) whose coordinates are computed from the wire width and the manufacturing grid.

Figure 4-9



The octagon is always symmetric about the x and y axis. The coordinates are computed to be exactly the same wire width as equivalent horizontal or vertical wire widths, and as close as possible for the diagonal widths (they are always slightly bigger because of rounding of irrational values), while forcing the coordinates to remain on the manufacturing grid. The wire width must be an even multiple of the manufacturing grid in order to keep A and B on the manufacturing grid.

Assume the following rules:

- W = wire width
- M = manufacturing grid (mgrid). This is derived from the LEF `MANUFACTURINGGRID` statement.
- D = diagonal width
- ceiling = round up the result to the nearest integer

The octagon coordinates are computed as:

$A = W/2$

$B = [\text{ceiling}(W/(\text{sqrt}(2) * M)\ ) * M] - A$

The derivation of B can be understood as:

$D = \text{sqrt}(\ (A + B)^2 + (A + B)^2)$ or $D = \text{sqrt}(2) * (A + B)$

The diagonal width (D) must be greater than or equal to the wire width (W), and B must be on the manufacturing grid, so D must be equal to W, which results in:

$D/\text{sqrt}(2) = A + B$

$B = D/\text{sqrt}(2) - A$ or $W/\text{sqrt}(2) - A$

To force B to be on the manufacturing grid, and keep the diagonal width greater than or equal to the wire width:

B on mgrid = $\text{ceiling}(B / M) * M$

Which results in the computation:

$B = [\text{ceiling}(W/\text{sqrt}(2) * M)\ ) * M] - A$

The following table lists examples coordinate computations:

Table 4-1

W = Width (µm)	M = mgrid (µm)	D = W/(sqrt(2)*M)	ceiling (D)	A (µm)	B (µm)	Diagonal width (µm)
1.0	0.005	141.42	142	0.5	0.21	1.0041
0.5	0.005	70.71	71	0.25	0.105	0.5020

0.15	0.005	21.21	22	0.075	0.035	0.1556
0.155*	0.005	21.92	22	0.0775*	0.0325*	0.1556
* A width of 0.155 is an odd multiple of the manufacturing grid and is not allowed because it would create coordinates for A and B that are off the manufacturing grid. It is shown for completeness to illustrate how the result is off grid.						

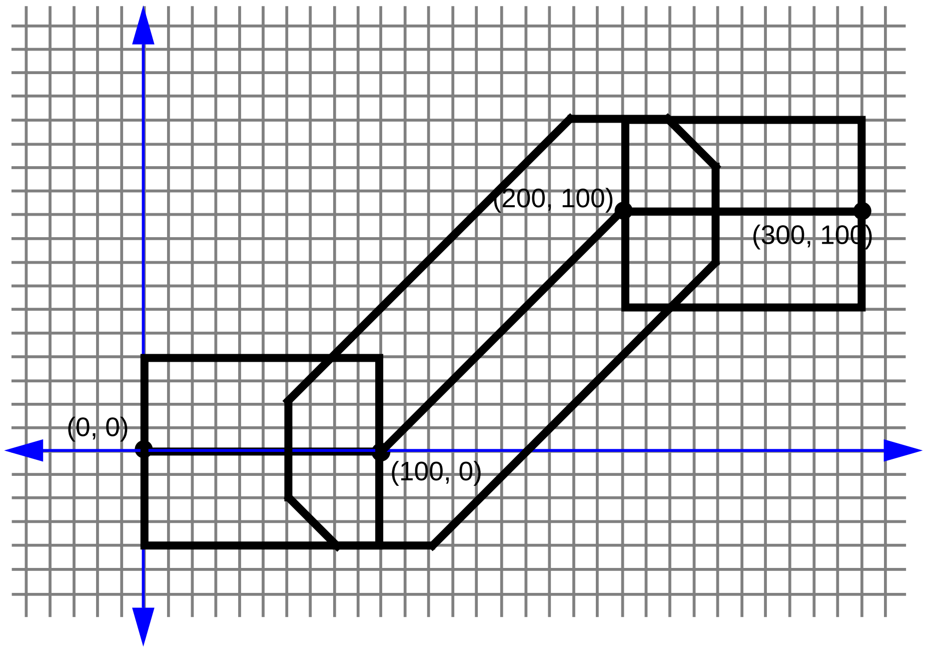
The default style only applies to 45-degree route segments; it does not apply to 90-degree route segments.

Example 1

The following two routes produce identical routing shapes, as shown in Figure 4-10 on page 202.

```
SPECIALNETS 1 ;
    - POWER (* power)
        + ROUTED metal3 80 ( 0 0 ) ( 100 0 ) ( 200 100 ) ( 300 100 ) ;
END SPECIALNETS

NETS 1 ;
    - mySignal ...  #mySignal uses the default routing rule width of 80
        + ROUTED metal3 ( 0 0 0 ) ( 100 0 0 ) ( 200 100 0 ) ( 300 100 0 ) ;
        #The wire extension was set to 0 for every point. The wire extension
        #is ignored for 45-degree route segments; the default octagon
        #overrides it.
END NETS
```

Figure 4-10

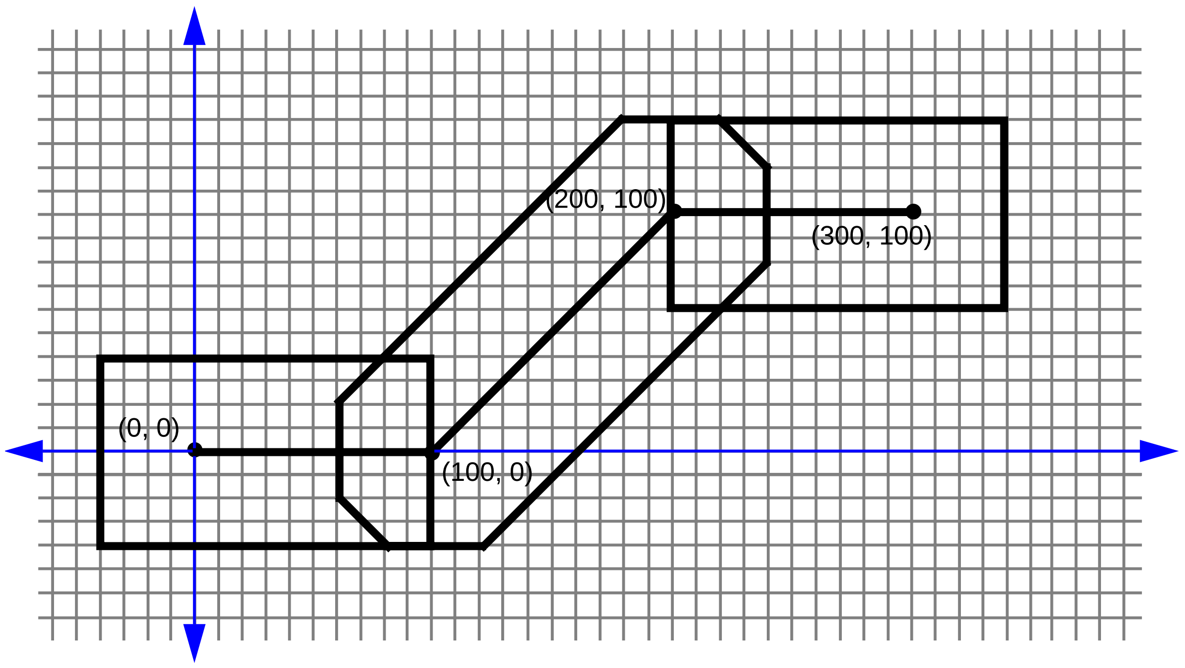


Example 2

The following regular route definition, using the traditional default wire extension of 1/2 * *width* for the first and last 90-degree endpoints, produces the route shown in Figure 4-11 on page 203.

```
SPECIALNETS 1;
    - POWER (* power)  #The half-width extensions are given for the first and last
        + ROUTED metal3 80 ( 0 0 40 ) ( 100 0 ) ( 200 100 ) ( 300 100 40 ) ;
           #The default extension is 0 for SPECIALNETS, so it is not given for
           #two middle points.
END SPECIALNETS

NETS 1 ;
    - mySignal ... #mySignal uses the default routing rule with width of 80
        + ROUTED metal3 ( 0 0 ) ( 100 0 0 ) ( 200 100 0 ) ( 300 100 ) ;
           #The default extension is half the width for NETS, so it is not
           #included for the first and last end-points.
END NETS
```

Figure 4-11

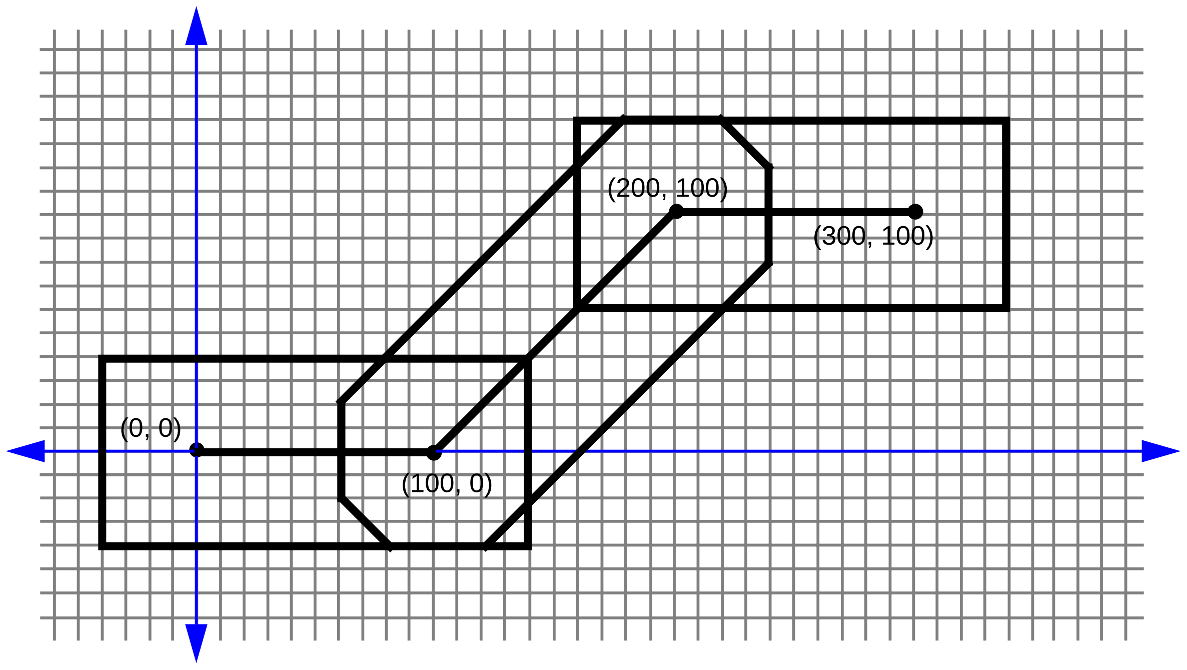


Example 3

The following definition, using the traditional default wire extension of 1/2 * *width* for all of the points, produces the route in Figure 4-12 on page 204.

```
SPECIALNETS 1 ;
    - POWER (* power)   #The half-width extensions are given explicitly
        + ROUTED metal3 80 ( 0 0 40 ) ( 100 40 ) ( 200 100 40 ) ( 300 100 40 ) ;
END SPECIALNETS

NETS 1 ;
    - mySignal ... #mySignal uses the default routing rule width of 80
        + ROUTED metal3 ( 0 0 ) ( 100 0 ) ( 200 100 ) ( 300 100 ) ;
           #All points use the implicit default 1/2 * width wire extensions.
END NETS
```

Figure 4-12



Technology

```
[TECHNOLOGY technologyName ;]
```

Specifies a technology name for the design in the database. In case of a conflict, the previous name remains in effect.

Tracks

```
[TRACKS
     [{X | Y} start DO numtracks STEP space
       [LAYER layerName] ...
     ;] ...]
```

Defines the routing grid for a standard cell-based design. Typically, the routing grid is generated when the floorplan is initialized. The first track is located at an offset from the placement grid set by the `OFFSET` value for the layer in the LEF file. The track spacing is the `PITCH` value for the layer defined in LEF.

`DO` *`numTracks`*	Specifies the number of tracks to create for the grid. You cannot specify 0 *`numtracks`*.

`LAYER` *`layerName`*	Specifies the routing layer used for the tracks. You can specify more than one layer.
`STEP` *`space`*	Specifies the spacing between the tracks.
`{X	Y}` *`start`*	Specifies the location and direction of the first track defined. `X` indicates vertical lines; `Y` indicates horizontal lines. *`start`* is the `X` or `Y` coordinate of the first line. For example, `X 3000` creates a set of vertical lines, with the first line going through (3000 0).

Units

```
[UNITS DISTANCE MICRONS dbuPerMicron ;]
```

Specifies the database units per micron (dbuPerMicron) to convert DEF distance units into microns.

LEF supports values of 100, 200, 1000, 2000, 10,000, and 20,000 for the LEF dbuPerMicron.

The following table shows the valid pairings of the LEF dbuPerMicron and the corresponding legal DEF dbuPerMicron values. The LEF dbuPerMicron must be greater than or equal to the DEF dbuPerMicron, otherwise you can get round-off errors.

LEF dbuPerMicron	Legal DEF dbuPerMicron
100	100
200	100, 200
1000	100, 200, 1000
2000	100, 200, 1000, 2000
10,000	100, 200, 1000, 2000, 10,000
20,000	100, 200, 1000, 2000, 10,000, 20,000

Using DEF Units

The following table shows examples of how DEF units are used:

Units	DEF Units	DEF Value Example	Real Value
Time	.001 nanosecond	1500	1.5 nanoseconds
Capacitance	.000001 picofarad	1,500,000	1.5 picofarads

Units	DEF Units	DEF Value Example	Real Value
Resistance	.ooo1 ohm	15,000	1.5 ohms
Power	.0001 milliwatt	15,000	1.5 milliwatts
Current	.0001 milliamp	15,000	1.5 milliamps
Voltage	.001 volt	1500	1.5 volts

The DEF reader assumes divisor factors such that DEF data is given in the database units shown below.

Unit	Database Precision
1 nanosecond	= 1000 DBUs
1 picofarad	= 1,000,000 DBUs
1 ohm	= 10,000 DBUs
1 milliwatt	= 10,000 DBUs
1 milliampere	= 10,000 DBUs
1 volt	= 1000 DBUs

Version

```
[VERSION versionNumber ;]
```

Specifies which version of the DEF syntax is being used.

Note: The `VERSION` statement is not required in a DEF file; however, you should specify it, because it prevents syntax errors caused by the inadvertent use of new versions of DEF with older tools that do not support the new version syntax.

Vias

```
[VIAS numVias ;
     [– viaName
        [   + VIARULE viaRuleName
               + CUTSIZE xSize ySize
               + LAYERS botmetalLayer cutLayer topMetalLayer
               + CUTSPACING xCutSpacing yCutSpacing
               + ENCLOSURE xBotEnc yBotEnc xTopEnc yTopEnc
               [+ ROWCOL numCutRows NumCutCols]
               [+ ORIGIN xOffset yOffset]
```

```
            [+ OFFSET xBotOffset yBotOffset xTopOffset yTopOffset]
            [+ PATTERN cutPattern] ]
      | [ + RECT layerName pt pt | + POLYGON layerName pt pt pt] ...]
    ;] ...
END VIAS]
```

Lists the names and geometry definitions of all generated vias in the design. Two types of vias can be listed: fixed vias and generated vias.

A fixed via is defined using rectangles or polygons, and does not use a `VIARULE`. The fixed via name must mean the same via in all associated LEF and DEF files.

A generated via is defined using `VIARULE` parameters to indicate that it was derived from a `VIARULE GENERATE` statement. For a generated via, the via name is only used locally inside this DEF file. The geometry and parameters are maintained, but the name can be freely changed by applications that use this via when writing out LEF and DEF files to avoid possible via name collisions with other DEF files.

`CUTSIZE` *`xSize ySize`* Specifies the required width (*`xSize`*) and height (*`ySize`*) of the cut layer rectangles.
Type: Integer, specified in DEF database units

`CUTSPACING` *`xCutSpacing yCutSpacing`*
Specifies the required x and y spacing between cuts. The spacing is measured from one cut edge to the next cut edge.
Type: Integer, specified in DEF database units

`ENCLOSURE` *`xBotEnc yBotEnc xTopEnc yTopEnc`*
Specifies the required x and y enclosure values for the bottom and top metal layers. The enclosure measures the distance from the cut array edge to the metal edge that encloses the cut array (see Figure 4-13 on page 211).
Type: Integer, specified in DEF database units

`LAYERS` *`botMetalLayer cutLayer TopMetalLayer`*
Specifies the required names of the bottom routing layer, cut layer, and top routing layer. These layer names must be previously defined in layer definitions, and must match the layer names defined in the specified LEF *`viaRuleName`*.

`numVias` Specifies the number of vias listed in the `VIA` statement.

`OFFSET` *`xBotOffset yBotOffset xTopOffset yTopOffset`*

Specifies the x and y offset for the bottom and top metal layers. These values allow each metal layer to be offset independently.

By default, the `0,0` origin of the via is the center of the cut array, and the enclosing metal rectangles. After the non-shifted via is computed, the metal layer rectangles are shifted by adding the appropriate values—the *`x/y BotOffset`* values to the metal layer below the cut layer, and the *`x/y TopOffset`* values to the metal layer above the cut layer.

These offset values are in addition to any offset caused by the `ORIGIN` values. For an example and illustration of this syntax, see Example 4-24 on page 210.
Default: `0`, for all values
Type: Integer, in DEF database units

`ORIGIN` *`xOffset yOffset`*

Specifies the x and y offset for all of the via shapes. By default, the `0,0` origin of the via is the center of the cut array, and the enclosing metal rectangles. After the non-shifted via is computed, all cut and metal rectangles are shifted by adding these values. For an example and illustration of this syntax, see Example 4-24 on page 210.
Default: `0`, for both values
Type: Integer, in DEF database units

`PATTERN` *`cutPattern`*

Specifies the cut pattern encoded as an ASCII string. This parameter is only required when some of the cuts are missing from the array of cuts, and defaults to "all cuts are present," if not specified.

For information on and examples of via cut patterns, see “Creating Via Cut Patterns” on page 212.

The *`cutPattern`* syntax is defined as follows:

```
numRows_rowDefinition
  [_numRows_rowDefinition] ...
```

`numRows`

Specifies a hexadecimal number that indicates how many times to repeat the following row definition. This number can be more than one digit.

`rowDefinition`	Defines one row of cuts, from left to right.

The *`rowDefinition`* syntax is defined as follows:

`{[R`*`repeatNumber`*`]`*`hexDigitCutPattern`*`} ...`

`hexDigitCutPattern`	Specifies a single hexadecimal digit that encodes a 4-bit binary value in which `1` indicates a cut is present, and `0` indicates a cut is not present.
`repeatNumber`	Specifies a single hexadecimal digit that indicates how many times to repeat *`hexDigitCutPattern`*.

`POLYGON` *`layerName pt pt pt`*

Defines the via geometry for the specified layer. You must specify at least three points to generate the polygon, and the edges must be parallel to the x axis, the y axis, or at a 45-degree angle. S

Each `POLYGON` statement defines a polygon generated by connecting each successive point, and then the first and last points. The *`pt`* syntax corresponds to a coordinate pair, such as (*x y*). Specify an asterisk (`*`) to repeat the same value as the previous *x* or *y* value from the last point.
Type: (*x y*) Integer, specified in database units

For example, `+ POLYGON ( 0 0 ) ( 10 10 ) ( 10 0 )` creates a triangle shape.

`RECT` *`layerName pt pt`*

Defines the via geometry for the specified layer. The points are specified with respect to the via origin. In most cases, the via origin is the center of the via bounding box. All geometries for the via, including the cut layers, are output by the DEF writer.

`ROWCOL` *`numCutRows numCutCols`*

Specifies the number of cut rows and columns that make up the cut array.
Default: `1`, for both values
Type: Positive integer, for both values

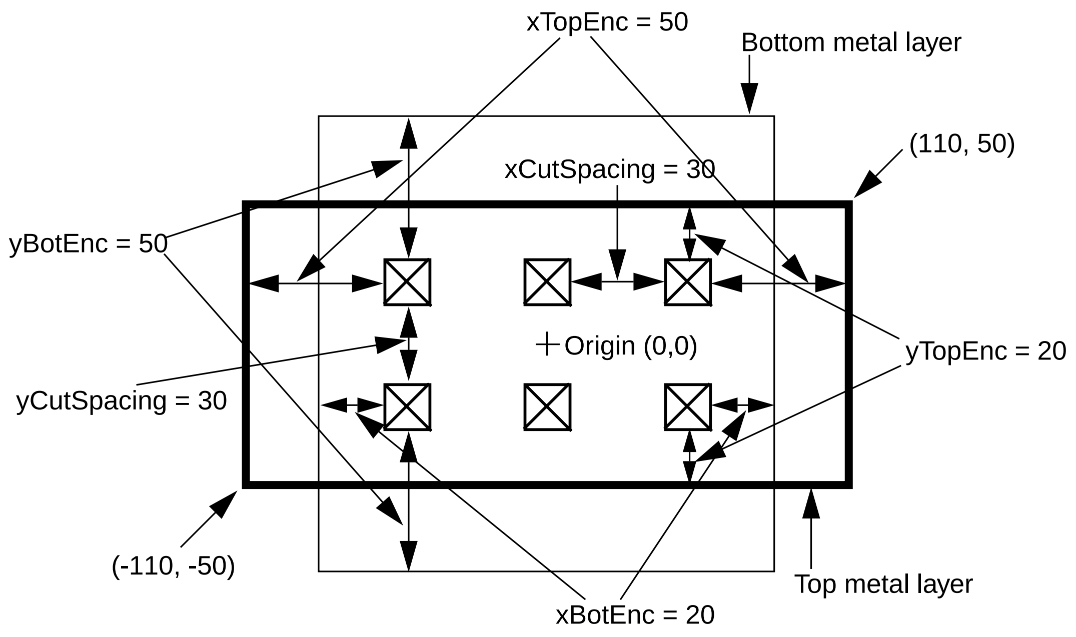
viaName	Specifies the via name. Via names are generated by appending a number after the rule name. Vias are numbered in the order in which they are created.
VIARULE *viaRuleName*	Specifies the name of the LEF VIARULE that produced this via. This name must be specified before you define any of the other parameters, and must refer to a VIARULE GENERATE via rule. It cannot refer to a VIARULE without a GENERATE keyword. Specifying the reserved via rule name of DEFAULT indicates that the via should use the previously defined VIARULE GENERATE rule with the DEFAULT keyword that exists for this routing-cut-routing layer combination. This makes it possible for a tool that does not use the LEF VIARULE technology section to still generate DEF generated-via parameters by using the default rule.

Example 4-24 Via Rules

The following via rule describes a non-shifted via (that is, a via with no OFFSET or ORIGIN parameters). There are two rows and three columns of via cuts. Figure 4-13 on page 211 illustrates this via rule.

```
    - myUnshiftedVia
        + VIARULE myViaRule
        + CUTSIZE 20 20             #xCutSize yCutSize
        + LAYERS metal1 cut12 metal2
        + CUTSPACING 30 30          #xCutSpacing yCutSpacing
        + ENCLOSURE 20 50 50 20     #xBotEnc yBotEnc xTopEnc yTopEnc
        + ROWCOL 2 3 ;
```

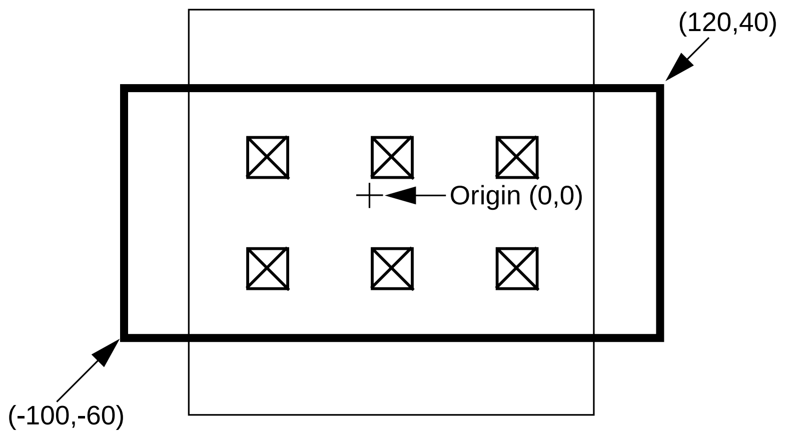
Figure 4-13 Via Rule



The same via rule with the following `ORIGIN` parameter shifts all of the metal and cut rectangles by 10 in the x direction, and by -10 in the y direction (see Figure 4-14 on page 211):

```
+ ORIGIN 10 -10
```

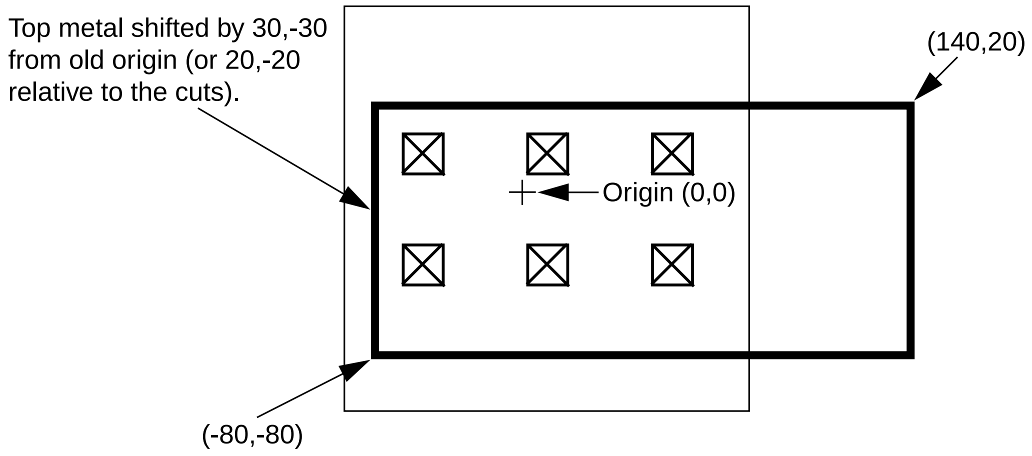
Figure 4-14 Via Rule With Origin



If the same via rule contains the following `ORIGIN` and `OFFSET` parameters, all of the rectangles shift by 10, -10. In addition, the top layer metal rectangle shifts by 20, -20, which means that the top metal shifts by a total of 30, -30.

```
    + ORIGIN 10 -10
    + OFFSET 0 0 20 -20
```

Figure 4-15 Via Rule With Origin and Offset



Creating Via Cut Patterns

Via cuts are defined as a series of rows, starting at the bottom, left corner. Each row definition defines one row of cuts, from left to right, and rows are numbered from bottom to top.

The `PATTERN` syntax that defines rows uses the `ROWCOL` parameters to specify the cut array. If the row has more bits than the *`numCutCols`* value in the `ROWCOL` parameter for this via, the last bits are ignored. The number of rows defined must equal the *`numCutRows`* value in the `ROWCOL` parameter.

Figure 4-16 on page 213 illustrates the following via cut pattern syntax:

```
    - myVia
        + VIARULE myViaRule
        ...
        + ROWCOL 5 5
        + PATTERN 2_F0_2_F8_1_78 ;]
```

The last three bits of `F0`, `F8`, and `78` are ignored because only five bits are allowed in a row. Therefore, the following `PATTERN` syntax gives the identical pattern:

```
    + PATTERN 2_F7_2_FF_1_7F
```

Figure 4-16

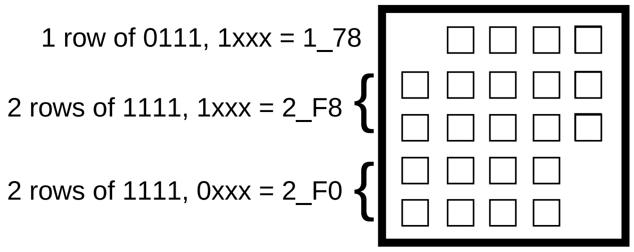
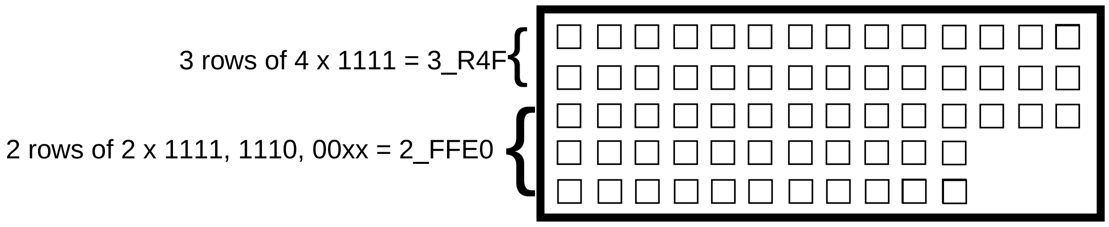

1 row of 0111, 1xxx = 1_78
2 rows of 1111, 1xxx = 2_F8
2 rows of 1111, 0xxx = 2_F0

Figure 4-17 on page 213 illustrates the following via cut pattern syntax:

```
- myVia
    + VIARULE myViaRule
    ...
    + ROWCOL 5 14
    + PATTERN 2_FFE0_3_R4F ;
```

The R4F value indicates a repeat of four Fs. The last two bits of each row definition are ignored because only 14 bits allowed in each row.

Figure 4-17


3 rows of 4 x 1111 = 3_R4F
2 rows of 2 x 1111, 1110, 00xx = 2_FFE0

A

Examples

This appendix contains information about the following topics.

LEF

```
# DEMO4 CHIP - 1280 ARRAY
NAMESCASESENSITIVE ON
    &alias &&area = (73600,74400) (238240,236400) &endalias
    &alias &&core = (85080,85500) (226760,224700) &endalias
    &alias &&m2stripes = sroute stripe net vss net vdd layer m2
        width
        320 count 2 pattern 87900 4200 218100
        area &&area core &&core &endalias
    &alias &&m3stripes = sroute stripe net vss net vdd layer m3
        width
        600 count 2 pattern 89840 6720 217520
        area &&area core &&core &endalias
    &alias &&powerfollowpins = sroute follow net vss net vdd layer
        m1 width 560
        area &&area core &&core &endalias
    &alias &&powerrepair = sroute repair net vss net vdd area
        &&area core &&core &endalias
# PLACEMENT SITE SECTION
SITE CORE1 SIZE 67.2 BY 6 ; # GCD of all Y sizes of Macros END
    CORE1
SITE IOX SIZE 37.8 BY 444 ; # 151.2 / 4 = 37.8 , 4 sites per pad END IOX
SITE IOY SIZE 436.8 BY 30 ; # 150 / 5 = 30 , 5 sites per pad END IOY
SITE SQUAREBLOCK SIZE 268.8 BY 252 ; END SQUAREBLOCK
SITE I2BLOCK SIZE 672 BY 504 ; END I2BLOCK
SITE LBLOCK SIZE 201.6 BY 168 ; END LBLOCK
SITE CORNER SIZE 436.8 BY 444 ; END CORNER
```

```
LAYER POLYS TYPE MASTERSLICE ; END POLYS
LAYER PW TYPE MASTERSLICE ; END PW
LAYER NW TYPE MASTERSLICE ; END NW
LAYER PD TYPE MASTERSLICE ; END PD
LAYER ND TYPE MASTERSLICE ; END ND
LAYER CUT01 TYPE CUT ; END CUT01
LAYER M1 TYPE ROUTING ; DIRECTION VERTICAL ; PITCH 5.6 ; WIDTH2.6 ;
    SPACING 1.5 ; END M1
LAYER CUT12 TYPE CUT ; END CUT12
LAYER M2 TYPE ROUTING ; DIRECTION HORIZONTAL ; PITCH 6.0 ;
    WIDTH 3.2 ;SPACING 1.6 ; END M2
LAYER CUT23 TYPE CUT ; END CUT23
LAYER M3 TYPE ROUTING ; DIRECTION VERTICAL ; PITCH 5.6 ; WIDTH 3.6;
    SPACING 1.6 ; END M3
LAYER OVERLAP TYPE OVERLAP ; END OVERLAP
VIA C2PW DEFAULT LAYER PW ; RECT -2.0 -2.0 2.0 2.0 ;
    LAYER CUT01 ; RECT -0.6 -0.6 0.6 0.6 ;
    LAYER M1 ; RECT -2.0 -2.0 2.0 2.0 ; END C2PW
VIA C2NW DEFAULT LAYER NW ; RECT -2.0 -2.0 2.0 2.0 ;
    LAYER CUT01 ; RECT -0.6 -0.6 0.6 0.6 ;
    LAYER M1 ; RECT -2.0 -2.0 2.0 2.0 ; END C2NW
VIA C2PD DEFAULT LAYER PD ; RECT -2.0 -2.0 2.0 2.0 ;
    LAYER CUT01 ; RECT -0.6 -0.6 0.6 0.6 ;
    LAYER M1 ; RECT -2.0 -2.0 2.0 2.0 ; END C2PD
VIA C2ND DEFAULT LAYER ND ; RECT -2.0 -2.0 2.0 2.0 ;
    LAYER CUT01 ; RECT -0.6 -0.6 0.6 0.6 ;
    LAYER M1 ; RECT -2.0 -2.0 2.0 2.0 ; END C2ND
VIA C2POLY DEFAULT LAYER POLYS ; RECT -2.0 -2.0 2.0 2.0 ;
    LAYER CUT01 ; RECT -0.6 -0.6 0.6 0.6 ;
    LAYER M1 ; RECT -2.0 -2.0 2.0 2.0 ; END C2POLY
VIA VIA12 DEFAULT LAYER M1 ; RECT -2.0 -2.0 2.0 2.0 ;
    LAYER CUT12 ; RECT -0.7 -0.7 0.7 0.7 ;
    LAYER M2 ; RECT -2.0 -2.0 2.0 2.0 ; END VIA12
VIA VIA23 DEFAULT LAYER M3 ; RECT -2.0 -2.0 2.0 2.0 ;
    LAYER CUT23 ; RECT -0.8 -0.8 0.8 0.8 ;
    LAYER M2 ; RECT -2.0 -2.0 2.0 2.0 ; END VIA23
    SPACING SAMENET CUT01 CUT12 4.0 ;
    SAMENET CUT12 CUT23 4.0 ; END SPACING
VIARULE TURN1 GENERATE LAYER M1 ; DIRECTION VERTICAL ;
    LAYER M1 ; DIRECTION HORIZONTAL ; END TURN1
VIARULE TURN2 GENERATE LAYER M2 ; DIRECTION VERTICAL ;
    LAYER M2 ; DIRECTION HORIZONTAL ; END TURN2
VIARULE TURN3 GENERATE LAYER M3 ; DIRECTION VERTICAL ;
    LAYER M3 ; DIRECTION HORIZONTAL ; END TURN3
VIA VIACENTER12 LAYER M1 ; RECT -4.6 -2.2 4.6 2.2 ;
    LAYER CUT12 ; RECT -3.1 -0.8 -1.9 0.8 ; RECT 1.9 -0.8 3.1 0.8 ;
    LAYER M2 ; RECT -4.4 -2.0 4.4 2.0 ; END VIACENTER12
```

```
VIA VIATOP12 LAYER M1 ; RECT -2.2 -2.2 2.2 8.2 ;
    LAYER CUT12 ; RECT -0.8 5.2 0.8 6.8 ;
    LAYER M2 ; RECT -2.2 -2.2 2.2 8.2 ; END VIATOP12
VIA VIABOTTOM12 LAYER M1 ; RECT -2.2 -8.2 2.2 2.2 ;
    LAYER CUT12 ; RECT -0.8 -6.8 0.8 -5.2 ;
    LAYER M2 ; RECT -2.2 -8.2 2.2 2.2 ; END VIABOTTOM12
VIA VIALEFT12 LAYER M1 ; RECT -7.8 -2.2 2.2 2.2 ;
    LAYER CUT12 ; RECT -6.4 -0.8 -4.8 0.8 ;
    LAYER M2 ; RECT -7.8 -2.2 2.2 2.2 ; END VIALEFT12
VIA VIARIGHT12 LAYER M1 ; RECT -2.2 -2.2 7.8 2.2 ;
    LAYER CUT12 ; RECT 4.8 -0.8 6.4 0.8 ;
    LAYER M2 ; RECT -2.2 -2.2 7.8 2.2 ; END VIARIGHT12
VIA VIABIGPOWER12 LAYER M1 ; RECT -21.0 -21.0 21.0 21.0 ;
    LAYER CUT12 ; RECT -2.4 -0.8 2.4 0.8 ;
        RECT -19.0 -19.0 -14.2 -17.4 ; RECT -19.0 17.4 -14.2
            19.0;
        RECT 14.2 -19.0 19.0 -17.4 ; RECT 14.2 17.4 19.0 19.0 ;
        RECT -19.0 -0.8 -14.2 0.8 ; RECT -2.4 -19.0 2.4 -17.4 ;
        RECT 14.2 -0.8 19 0.8 ; RECT -2.4 17.4 2.4 19.0 ;
    LAYER M2 ; RECT -21.0 -21.0 21.0 21.0 ; END VIABIGPOWER12
VIARULE VIALIST12 LAYER M1 ; DIRECTION VERTICAL ; WIDTH 9.0 TO
    9.6;
    LAYER M2 ; DIRECTION HORIZONTAL ; WIDTH 3.0 TO 3.0 ;
        VIA VIACENTER12 ; VIA VIATOP12 ; VIA VIABOTTOM12 ;
        VIA VIALEFT12 ; VIA VIARIGHT12 ; END VIALIST12
VIARULE VIAGEN12 GENERATE LAYER M1 ; DIRECTION VERTICAL ;
    OVERHANG 1.4 ;
    LAYER M2 ; DIRECTION HORIZONTAL ; OVERHANG 1.4 ;
    LAYER CUT12 ; RECT -0.8 -0.8 0.8 0.8 ;
    SPACING 5.6 BY 6.0 ; END VIAGEN12
VIA VIACENTER23 LAYER M3 ; RECT -2.2 -2.2 2.2 2.2 ;
    LAYER CUT23 ; RECT -0.8 -0.8 0.8 0.8 ;
    LAYER M2 ; RECT -2.0 -2.0 2.0 2.0 ; END VIACENTER23
VIA VIATOP23 LAYER M3 ; RECT -2.2 -2.2 2.2 8.2 ;
    LAYER CUT23 ; RECT -0.8 5.2 0.8 6.8 ;
    LAYER M2 ; RECT -2.2 -2.2 2.2 8.2 ; END VIATOP23
VIA VIABOTTOM23 LAYER M3 ; RECT -2.2 -8.2 2.2 2.2 ;
    LAYER CUT23 ; RECT -0.8 -6.8 0.8 -5.2 ;
    LAYER M2 ; RECT -2.2 -8.2 2.2 2.2 ; END VIABOTTOM23
VIA VIALEFT23 LAYER M3 ; RECT -7.8 -2.2 2.2 2.2 ;
    LAYER CUT23 ; RECT -6.4 -0.8 -4.8 0.8 ;
    LAYER M2 ; RECT -7.8 -2.2 2.2 2.2 ; END VIALEFT23
VIA VIARIGHT23 LAYER M3 ; RECT -2.2 -2.2 7.8 2.2 ;
    LAYER CUT23 ; RECT 4.8 -0.8 6.4 0.8 ;
    LAYER M2 ; RECT -2.2 -2.2 7.8 2.2 ; END VIARIGHT23
VIARULE VIALIST23 LAYER M3 ; DIRECTION VERTICAL ; WIDTH 3.6 TO
    3.6 ;
```

```
        LAYER M2 ; DIRECTION HORIZONTAL ; WIDTH 3.0 TO 3.0 ;
            VIA VIACENTER23 ; VIA VIATOP23 ; VIA VIABOTTOM23 ;
            VIA VIALEFT23 ; VIA VIARIGHT23 ; END VIALIST23
VIARULE VIAGEN23 GENERATE LAYER M3 ; DIRECTION VERTICAL ;
    OVERHANG 1.4 ;
    LAYER M2 ; DIRECTION HORIZONTAL ; OVERHANG 1.4 ;
    LAYER CUT23 ; RECT -0.8 -0.8 0.8 0.8 ;
    SPACING 5.6 BY 6.0 ; END VIAGEN23


MACRO CORNER CLASS ENDCAP BOTTOMLEFT ; SIZE 436.8 BY 444 ; SYMMETRY X
Y ; SITE CORNER ;
    PIN VDD SHAPE RING ; DIRECTION INOUT ;
        PORT LAYER M2 ; WIDTH 20 ; PATH 426.8 200 200 200 200 434 ;
            END END VDD
    PIN VSS SHAPE RING ; DIRECTION INOUT ;
        PORT LAYER M2 ; WIDTH 20 ; PATH 100 434 100 100 ; LAYER M1;
            WIDTH 20 ; PATH 100 100 426.8 100 ; END END VSS END
                CORNER
MACRO IN1X class pad ; FOREIGN IN1X ; SIZE 151.2 BY 444 ;
    SYMMETRY X ; SITE IOX ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M1 ; PATH 61.6 444 72.8 444 ; END END Z
    PIN PO DIRECTION OUTPUT ;
        PORT LAYER M1 ; PATH 78.4 444 84.0 444 ; END END PO
    PIN A DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 95.2 444 100.8 444 ; END END A
    PIN PI DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 106.4 444 112 444 ; END END PI
    PIN VDD DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M2 ; WIDTH 20 ; PATH 10 200 141.2 200 ; END
    END VDD
    PIN VSS DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M1 ; WIDTH 20 ; PATH 10 100 141.2 100 ; END
        END VSS END IN1X
MACRO IN1Y EEQ IN1X ; FOREIGN IN1Y ; class pad ;SIZE 436.8 BY 150 ;
    SYMMETRY Y ; SITE IOY ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 0 69 0 75 ; END END Z
    PIN PO DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 0 81 0 87 ; END END PO
    PIN A DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 0 51 0 57 ; ENDEND A
    PIN PI DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 0 39 0 45 ; END END PI
    PIN VDD DIRECTION INOUT ; SHAPE ABUTMENT ;
```

```
        PORT LAYER M2 ; WIDTH 20 ; PATH 236.8 10 236.8 140 ; END
    END VDD
    PIN VSS DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M2 ; WIDTH 20 ; PATH 336.8 10 336.8 140 ; END
        END VSS END IN1Y
MACRO FILLER FOREIGN FILLER ; SIZE 67.2 BY 6 ; SYMMETRY X Y R90;
    SITE CORE1 ;
    PIN VDD DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M1 ; RECT 45.8 0 55 6 ; END END VDD
    PIN VSS DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M1 ; RECT 12.2 0 21.4 6 ; END END VSS
    OBS LAYER M1 ; RECT 24.1 1.5 43.5 4.5 ; END END FILLER
MACRO INV FOREIGN INVS ; SIZE 67.2 BY 24 ; SYMMETRY X Y ; SITE CORE1 ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 30.8 9 42 9 ; END END Z
    PIN A DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 25.2 15 ; END END A
    PIN VDD DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 50.4 4.6 50.4 13.4 ; END
    END VDD
    PIN VSS DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 16.8 4.6 16.8 13.4 ; END
    END VSS
    OBS LAYER M1 ; RECT 24.1 1.5 43.5 16.5 ; END END INV
MACRO BUF FOREIGN BUFS ; SIZE 67.2 BY 126 ; SYMMETRY X Y ; SITE
    CORE1 ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 25.2 39 42 39 ; END END Z
    PIN A DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 30.8 33 ; END END A
    PIN VDD DIRECTION INOUT ; SHAPE FEEDTHRU ;
        PORT LAYER M1 ; WIDTH 5.6 ;
        PATH 50.4 4.6 50.4 10.0 56.0 10.0 56.0 115.8 50.4 115.8
            50.4 121.4 ; END END VDD
    PIN VSS DIRECTION INOUT ; SHAPE FEEDTHRU ;
        PORT LAYER M1 ; WIDTH 5.6 ;
        PATH 16.8 4.6 16.8 10.0 11.2 10.0 11.2 115.8 16.8 115.8
            16.8 121.4 ; END END VSS
    OBS LAYER M1 ; RECT 24.1 1.5 43.5 124.5 ; END END BUF
MACRO BIDIR1X FOREIGN BIDIR1X ; class pad ; SIZE 151.2 BY 444 ;
    SYMMETRY X ; SITE IOX ;
    PIN IO DIRECTION INOUT ;
        PORT LAYER M1 ; PATH 61.6 444 67.2 444 ; END END IO
    PIN ZI DIRECTION OUTPUT ;
        PORT LAYER M1 ; PATH 78.4 444 84.0 444 ; END END ZI
    PIN PO DIRECTION OUTPUT ;
        PORT LAYER M1 ; PATH 95.2 444 100.8 444 ; END END PO
    PIN A DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 106.4 444 112.0 444 ; END END A
    PIN EN DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 134.4 444 140.0 444 ; END END EN
    PIN TN DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 28.0 444 33.6 444 ; END END TN
    PIN PI DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 44.8 444 50.4 444 ; END END PI
    PIN VDD DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M2 ; WIDTH 20 ; PATH 10 200 141.2 200 ; END END VDD
    PIN VSS DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M1 ; WIDTH 20 ; PATH 10 100 141.2 100 ; END
        END VSS END BIDIR1X

MACRO BIDIR1Y EEQ BIDIR1X ; class pad ; FOREIGN BIDIR1Y ; SIZE 436.8
    BY 150 ; SYMMETRY Y ; SITE IOY ;
    PIN IO DIRECTION INOUT ;
        PORT LAYER M2 ; PATH 0 69 0 75 ; END END IO
    PIN ZI DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 0 93 0 99 ; END END ZI
    PIN PO DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 0 81 0 87 ; END END PO
    PIN A DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 0 15 0 21 ; END END A
    PIN EN DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 0 27 0 33 ; END END EN
    PIN TN DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 0 39 0 45 ; END END TN
    PIN PI DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 0 51 0 57 ; END END PI
    PIN VDD DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M2 ; WIDTH 20 ; PATH 236.8 10 236.8 140 ; END
    END VDD
    PIN VSS DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M2 ; WIDTH 20 ; PATH 336.8 10 336.8 140 ; END
        END VSS END BIDIR1Y

MACRO OR2 FOREIGN OR2S ; SIZE 67.2 BY 42 ; SYMMETRY X Y ; SITE
    CORE1 ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 25.2 39 42 39 ; END END Z
    PIN A DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 25.2 15 ; END END A
    PIN B DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 25.2 3 ; END END B
    PIN VDD DIRECTION INOUT ; SHAPE FEEDTHRU ;
```

```
        PORT LAYER M1 ; WIDTH 5.6 ;
        PATH 50.4 4.6 50.4 10.0 ; PATH 50.4 27.4 50.4 37.4 ;
        VIA 50.4 3 C2PW ; VIA 50.4 21 C2PW ; VIA 50.4 33 C2PW ;
        VIA 50.4 39 C2PW ; END END VDD
    PIN VSS DIRECTION INOUT ; SHAPE FEEDTHRU ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 16.8 4.6 16.8 10.0 ;
        PATH 16.8 27.4 16.8 37.4 ;
        VIA 16.8 3 C2NW ; VIA 16.8 15 C2NW ; VIA 16.8 21 C2NW ;
        VIA 16.8 33 C2NW ; VIA 16.8 39 C2NW ; END END VSS
    OBS LAYER M1 ; RECT 24.1 1.5 43.5 40.5 ; END END OR2

MACRO AND2 FOREIGN AND2S ; SIZE 67.2 BY 84 ; SYMMETRY X Y ; SITE
    CORE1 ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 25.2 39 42 39 ; END END Z
    PIN A DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 42 15 ; END END A
    PIN B DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 42 3 ; END END B
    PIN VDD DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 50.4 4.6 50.4 79.4 ; END
    END VDD
    PIN VSS DIRECTION INOUT ; SHAPE ABUTMENT ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 16.8 4.6 16.8 79.4 ; END
        END VSS
    OBS LAYER M1 ; RECT 24.1 1.5 43.5 82.5 ; END END AND2

MACRO DFF3 FOREIGN DFF3S ; SIZE 67.2 BY 210 ; SYMMETRY X Y ; SITE
    CORE1 ;
    PIN Q DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 19.6 99 47.6 99 ; END END Q
    PIN QN DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 25.2 123 42 123 ; END END QN
    PIN D DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 30.8 51 ; END END D
    PIN G DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 25.2 3 ; END END G
    PIN CD DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 36.4 75 ; END END CD
    PIN VDD DIRECTION INOUT ; SHAPE FEEDTHRU ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 50.4 4.6 50.4 205.4 ;
    END END VDD
    PIN VSS DIRECTION INOUT ; SHAPE FEEDTHRU ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 16.8 4.6 16.8 205.4 ;
    END END VSS
    OBS LAYER M1 ; RECT 24.1 1.5 43.5 208.5 ; PATH 8.4 3 8.4 123 ;
        PATH 58.8 3 58.8 123 ; PATH 64.4 3 64.4 123; END END DFF3
```

```
MACRO NOR2 FOREIGN NOR2S ; SIZE 67.2 BY 42 ; SYMMETRY X Y ; SITE
    CORE1 ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M1 ; PATH 42 33 ; END END Z
    PIN A DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 25.2 15 ; END END A
    PIN B DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 36.4 9 ; END END B
    PIN VDD DIRECTION INOUT ; SHAPE FEEDTHRU ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 50.4 4.6 50.4 37.4 ; END
    END VDD
    PIN VSS DIRECTION INOUT ; SHAPE FEEDTHRU ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 16.8 4.6 16.8 37.4 ; END
    END VSS
    OBS LAYER M1 ; RECT 24.1 1.5 43.5 40.5 ; END END NOR2
MACRO AND2J EEQ AND2 ;FOREIGN AND2SJ ; SIZE 67.2 BY 48 ;
    SYMMETRY X Y ; ORIGIN 0 6 ; SITE CORE1 ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 25.2 33 42 33 ; END END Z
    PIN A DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 42 15 ; END END A
    PIN B DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 42 3 ; END END B
    PIN VDD DIRECTION INOUT ; SHAPE FEEDTHRU ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 50.4 -1.4 50.4 37.4 ;
    END END VDD
    PIN VSS DIRECTION INOUT ; SHAPE FEEDTHRU ;
        PORT LAYER M1 ; WIDTH 5.6 ; PATH 16.8 -1.4 16.8 37.4 ;
    END END VSS
    OBS LAYER M1 ; RECT 24.1 1.5 43.5 34.5 ; END END AND2J
MACRO SQUAREBLOCK FOREIGN SQUAREBLOCKS ; CLASS RING ;SIZE 268.8
    BY 252 ; SITE SQUAREBLOCK ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 22.8 21 246.0 21 ; END END Z
    PIN A DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 64.4 33 137.2 33 ;
        PATH 137.2 33 137.2 69 ; PATH 137.2 69 204.4 69 ; END
    END A
    PIN B DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 22.8 129 246.0 129 ; END END B
    PIN C DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 70 165 70 153 ; PATH 70 153 126 153 ;
        END END C
    PIN D DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 22.8 75 64.4 75 ; END END D
    PIN E DIRECTION INPUT ;
```

```
        PORT LAYER M2 ; PATH 22.8 87 64.4 87 ; END END E
    PIN F DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 22.8 99 64.4 99 ; END END F
    PIN G DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 22.8 111 64.4 111 ; END END G
    PIN VDD DIRECTION INOUT ; SHAPE RING ;
        PORT LAYER M1 ; WIDTH 3.6 ; PATH 4.0 3.5 4.0 248 ;
        PATH 264.8 100 264.8 248 ; PATH 150 3.5 150 100 ;
        LAYER M2 ; WIDTH 3.6 ; PATH 4.0 3.5 150 3.5 ;
        PATH 150 100 264.8 100 ; PATH 4.0 248 264.8 248 ; END
        END VDD
    PIN VSS DIRECTION INOUT ; SHAPE RING ;
        PORT LAYER M1 ; WIDTH 3.6 ; PATH 10 10 10 150 ;
        PATH 100 150 100 200 ; PATH 50 200 50 242 ;
        PATH 258.8 10 258.8 242 ; LAYER M2 ; WIDTH 3.6 ;
        PATH 10 150 100 150 ; PATH 100 200 50 200 ;
        PATH 10 10 258.8 10 ; PATH 50 242 258.8 242 ; END END VSS
    OBS LAYER M1 ; RECT 13.8 14.0 255.0 237.2 ; END
        END SQUAREBLOCK

MACRO I2BLOCK FOREIGN I2BLOCKS ; CLASS RING ; SIZE 672 BY 504 ;
    SITE I2BLOCK ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 22.8 21 649.2 21 ; END END Z
    PIN A DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 22.8 63 154.0 63 ; PATH 154.0 63 154.0
    129;
        PATH 154.0 129 447.6 129 ; END END A
    PIN B DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 137.2 423 447.6 423 ; END END B
    PIN C DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 204.4 165 271.6 165 ; END END C
    PIN D DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 204.4 171 271.6 171 ; END END D
    PIN E DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 204.4 213 204.4 213 ; END END E
    PIN F DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 406 249 406 273 ; END END F
    PIN G DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 338.8 249 338.8 273 ; END END G
    PIN H DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 372.4 357 372.4 381 ; END END H
    PIN VDD DIRECTION INOUT ; SHAPE RING ;
        PORT LAYER M1 ; WIDTH 3.6 ; PATH 668 3.5 668 80.5 ;
        PATH 467 80.5 467 465.5 ; PATH 668 465.5 668 500.5 ;
        PATH 4 500.5 4 465.5 ; PATH 138 465.5 138 80.5 ;
        PATH 4 80.5 4 3.5 ; LAYER M2 ; WIDTH 3.6 ; PATH 4 3.5 668 3.5;
        PATH 668 80.5 467 80.5 ; PATH 467 465.5 668 465.5 ;
        PATH 668 500.5 4 500.5 ; PATH 4 465.5 138 465.5 ;
        PATH 138 80.5 4 80.5 ; END END VDD
    PIN VSS DIRECTION INOUT ; SHAPE RING ;
        PORT LAYER M1 ; WIDTH 3.6 ; PATH 662 10 662 74 ;
        PATH 461 74 461 472 ; PATH 662 472 662 494 ; PATH 10 494 10
            472;
        PATH 144 472 144 74 ; PATH 10 74 10 10 ;LAYER M2 ; WIDTH
            3.6 ;
        PATH 10 10 662 10 ; PATH 662 74 461 74 ; PATH 461 472 662
            472 ;
        PATH 662 494 10 494 ; PATH 10 472 144 472 ; PATH 144 74 10
            74 ;
        END END VSS
    OBS LAYER M1 ; RECT 14 14 658 70 ; RECT 14 476 658 490 ;
        RECT 148 14 457 490 ; # rectilinear shape description
        LAYER OVERLAP ; RECT 0 0 672 84 ; RECT 134.4 84 470.4 462 ;
        RECT 0 462 672 504 ; END END I2BLOCK

MACRO LBLOCK FOREIGN LBLOCKS ; CLASS RING ; SIZE 201.6 BY 168 ; SITE
    LBLOCK ;
    PIN Z DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 2.8 15 198.8 15 ; END END Z
    PIN A DIRECTION OUTPUT ;
        PORT LAYER M2 ; PATH 2.8 81 137.2 81 ; PATH 137.2 81 137.2
            69 ;
        PATH 137.2 69 198.8 69 ; END END A
    PIN B DIRECTION INPUT ;
        PORT LAYER M2 ; PATH 2.8 165 64.4 165 ; END END B
    PIN C DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 2.8 93 2.8 105 ; END END C
    PIN D DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 64.4 93 64.4 105 ; END END D
    PIN E DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 198.8 39 198.8 39 ; END END E
    PIN F DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 198.8 45 198.8 45 ; END END F
    PIN G DIRECTION INPUT ;
        PORT LAYER M1 ; PATH 2.8 111 2.8 111 ; END END G
        PORT LAYER M2 ; WIDTH 3.6 ; PATH 1.8 27 199.8 27 ; END END VDD
    PIN VSS DIRECTION INOUT ;
        PORT LAYER M2 ; WIDTH 3.6 ; PATH 1.8 57 199.8 57 ; END END VSS
    OBS LAYER M2 ; RECT 1.0 80 66.2 166.5 ; RECT 1.0 1.5 200.6 23 ;
        RECT 1.0 31 200.6 53 ; RECT 1.0 61 200.6 82.5 ;
        # rectilinear shape description
        LAYER OVERLAP ; RECT 0 0 201.6 84 ; RECT 0 84 67.2 168 ;
        END END LBLOCK
END LIBRARY
```

DEF

The following example shows a design netlist.

```
DESIGN DEMO4CHIP ;
TECHNOLOGY DEMO4CHIP ;
ARRAY DEMO4 ;
UNITS DISTANCE MICRONS 100 ;
COMPONENTS 243 ;
```

```
- CORNER1 CORNER ; - CORNER2 CORNER ; - CORNER3 CORNER ;
- CORNER4 CORNER ; - C01 IN1X ; - C02 IN1Y ; - C04 IN1X ;
- C05 IN1X ; - C06 IN1Y ;
- C07 IN1Y ; - C08 IN1Y ; - C09 IN1Y ; - C10 IN1X ; - C11 IN1X ;
- C13 BIDIR1Y ; - C14 INV ; - C15 BUF ; - C16 BUF ; - C17 BUF ;
- C19 BIDIR1Y ; - C20 INV ; - C21 BUF ; - C22 BUF ; - C23 BUF ;
- C25 BIDIR1Y ; - C26 INV ; - C27 BUF ; - C28 BUF ; - C29 BUF ;
- C31 BIDIR1Y ; - C32 INV ; - C33 BUF ; - C34 BUF ; - C35 BUF ;
- C37 BIDIR1X ; - C39 INV ; - C40 BUF ; - C41 BUF ; - C42 BUF ;
- C44 BIDIR1X ; - C45 INV ; - C46 BUF ; - C47 BUF ; - C48 BUF ;
- C50 BIDIR1Y ; - C51 INV ; - C52 BUF ; - C53 BUF ; - C54 BUF ;
- C56 BIDIR1X ; - C57 INV ; - C58 BUF ; - C59 BUF ; - C60 BUF ;
- D02 BIDIR1X ; - D03 INV ; - D04 BUF ; - D05 BUF ; - D06 BUF ;
- D08 BIDIR1X ; - D09 INV ; - D10 BUF ; - D11 BUF ; - D12 BUF ;
- D14 BIDIR1X ; - D15 INV ; - D16 BUF ; - D17 BUF ; - D19 BUF ;
- D33 BIDIR1Y ; - D34 INV ; - D35 BUF ; - D36 BUF ; - D37 BUF ;
- D39 BIDIR1Y ; - D40 INV ; - D41 BUF ; - D42 BUF ; - D43 BUF ;
- D45 BIDIR1Y ; - D46 INV ; - D47 BUF ; - D48 BUF ; - D49 BUF ;
- D82 OR2 ; - D83 OR2 ; - D84 OR2 ; - D85 OR2 ; - D86 OR2 ;
- D87 OR2 ; - D88 OR2 ; - D89 OR2 ; - D90 OR2 ; - D91 OR2 ;
- D92 OR2 ; - D93 OR2 ;
- E01 AND3 ; - E02 AND3 ; - E03 AND3 ; - E04 AND3 ; - E05 AND3 ;
- E06 AND3 ; - E07 AND3 ; - E08 AND3 ; - E09 AND3 ; - E10 AND3 ;
- E11 AND3 ; - E12 AND3 ; - E13 AND3 ; - E14 AND3 ; - E15 AND3 ;
- E16 AND3 ;
- EE16 IN1X ; - E17 IN1X ; - E18 IN1X ; - E19 IN1X ; - E20 IN1X ;
- E21 IN1X ; - E22 IN1X ; - E23 IN1Y ; - E24 IN1Y ; - E25 IN1Y ;
- E26 INV ; - E27 AND2 ; - E28 AND2 ; - E29 AND2 ; - E30 AND2 ;
- E31 AND2 ; - E32 AND2 ; - E33 OR2 ; - E34 OR2 ; - E35 OR2 ;
- E36 OR2 ; - E37 IN1Y ; - E38A01 DFF3 ; - E38A02 DFF3 ;
- E38A03 DFF3 ;- E38A04 DFF3 ; - E38A05 DFF3 ; - F01 I2BLOCK ;
- F04 OR2 ; - F06 OR2 ; - F07 OR2 ; - F08 OR2 ; - F09 SQUAREBLOCK ;
- F12 LBLOCK ;
- Z14 INV ; - Z15 BUF ; - Z16 BUF ; - Z17 BUF ; - Z20 INV ;
- Z21 BUF ; - Z22 BUF ; - Z23 BUF ; - Z26 INV ; - Z27 BUF ;
- Z28 BUF ; - Z29 BUF ; - Z32 INV ; - Z33 BUF ; - Z34 BUF ;
- Z35 BUF ; - Z39 INV ; - Z40 BUF ; - Z41 BUF ; - Z42 BUF ;
- Z45 INV ; - Z46 BUF ; - Z47 BUF ; - Z48 BUF ; - Z51 INV ;
- Z52 BUF ; - Z53 BUF ; - Z54 BUF ; - Z57 INV ; - Z58 BUF ; - Z59 BUF ;
- Z60 BUF ; - Z103 INV ; - Z104 BUF ; - Z105 BUF ; - Z106 BUF ;
- Z109 INV ; - Z110 BUF ; - Z111 BUF ; - Z112 BUF ; - Z115 INV ;
- Z116 BUF ; - Z117 BUF ; - Z119 BUF ; - Z134 INV ; - Z135 BUF ;
- Z136 BUF ; - Z137 BUF ; - Z140 INV ; - Z141 BUF ; - Z142 BUF ;
- Z143 BUF ; - Z146 INV ; - Z147 BUF ; - Z148 BUF ; - Z149 BUF ;
- Z182 OR2 ; - Z183 OR2 ; - Z184 OR2 ; - Z185 OR2 ; - Z186 OR2 ;
- Z187 OR2 ; - Z188 OR2 ; - Z189 OR2 ; - Z190 OR2 ; - Z191 OR2 ;
- Z192 OR2 ; - Z193 OR2 ; - Z201 AND3 ; - Z202 AND3 ; - Z203 AND3 ;
- Z204 AND3 ; - Z205 AND3 ; - Z206 AND3 ; - Z207 AND3 ; - Z208 AND3 ;
- Z209 AND3 ; - Z210 AND3 ; - Z211 AND3 ; - Z212 AND3 ; - Z213 AND3 ;
- Z214 AND3 ; - Z215 AND3 ; - Z216 AND3 ; - Z226 INV ; - Z227 AND2 ;
- Z228 AND2 ; - Z229 AND2 ; - Z230 AND2 ; - Z231 AND2 ; - Z232 AND2 ;
- Z233 OR2 ; - Z234 OR2 ; - Z235 OR2 ; - Z236 OR2 ; - Z38A01 DFF3 ;
- Z38A02 DFF3 ; - Z38A03 DFF3 ; - Z38A04 DFF3 ; - Z38A05 DFF3 ;
END COMPONENTS

NETS 222 ;
- VDD ( Z216 B ) ( Z215 B ) ( Z214 C ) ( Z214 B )
( Z213 C ) ( Z213 B ) ( Z212 C ) ( Z212 B ) ( Z211 C ) ( Z211 B )
( Z210 C ) ( E23 Z ) ( Z143 Z ) ( Z142 Z ) ( Z141 Z ) ( Z119 Z )
( Z117 Z ) ( Z116 Z ) ( Z106 Z ) ( Z105 Z ) ( Z104 Z ) ( Z34 Z )
( Z33 Z ) ( Z28 Z ) ( Z27 Z ) ( Z22 Z ) ( Z21 Z ) ( Z16 Z )
( Z15 Z ) ( D45 PO ) ( D14 PO ) ( C01 PI ) ( D45 TN ) ( D39 TN )
( D33 TN ) ( D14 TN ) ( D08 TN ) ( D02 TN ) ( C56 TN ) ( C50 TN )
( C44 TN ) ( C37 TN ) ( C31 TN ) ( C25 TN ) ( C19 TN ) ( C13 TN ) ;
- VSS ( Z209 C ) ( Z208 C ) ( Z207 C ) ( Z206 C ) ( Z205 C )
( Z204 C ) ( Z203 C ) ( Z202 C ) ( Z201 C ) ( Z149 Z ) ( Z148 Z )
( Z147 Z ) ( Z137 Z ) ( Z136 Z ) ( Z135 Z ) ( Z112 Z ) ( Z111 Z )
( Z110 Z ) ( Z60 Z ) ( Z59 Z ) ( Z58 Z ) ( Z54 Z ) ( Z53 Z )
( Z52 Z ) ( Z47 Z ) ( Z46 Z ) ( Z41 Z ) ( Z40 Z ) ( E18 Z )
( D49 Z ) ( D43 Z ) ( D45 A ) ( D39 A ) ( D33 A ) ( D14 A )
( D08 A ) ( D02 A ) ( C56 A ) ( C50 A ) ( C44 A ) ( C37 A )
( C31 A ) ( C25 A ) ( C19 A ) ( C13 A ) ; - XX1001 ( Z38A04 G )
( Z38A02 G ) ; - XX100 ( Z38A05 G ) ( Z38A03 G ) ( Z38A01 G ) ;
- XX907 ( Z236 B ) ( Z235 B ) ; - XX906 ( Z234 B ) ( Z233 B ) ;
- XX904 ( Z232 B ) ( Z231 B ) ; - XX903 ( Z230 B ) ( Z229 B ) ;
- XX902 ( Z228 B ) ( Z227 B ) ;
- XX900 ( Z235 A ) ( Z233 A ) ( Z232 A ) ( Z230 A ) ( Z228 A ) ( Z226 A ) ;
- Z38QN4 ( Z38A04 QN ) ( Z210 B ) ; - COZ131 ( Z38A04 Q ) ( Z210 A ) ;
- Z38QN3 ( Z38A03 QN ) ( Z209 B ) ; - COZ121 ( Z38A03 Q ) ( Z209 A ) ;
- Z38QN2 ( Z38A02 QN ) ( Z208 B ) ; - COZ111 ( Z38A02 Q ) ( Z208 A ) ;
- Z38QN1 ( Z38A01 QN ) ( Z207 B ) ; - COZ101 ( Z38A01 Q ) ( Z207 A ) ;
- XX901 ( Z236 A ) ( Z234 A ) ( Z231 A ) ( Z229 A ) ( Z227 A ) ( Z226 Z )
    ( Z193 A ) ;
- X415 ( Z149 A ) ( Z148 A ) ( Z147 A ) ( Z146 Z ) ; - X413 ( Z143 A )
    ( Z142 A ) ( Z141 A ) ( Z140 Z ) ;
- X411 ( Z137 A ) ( Z136 A ) ( Z135 A ) ( Z134 Z ) ;
- X405 ( Z119 A ) ( Z117 A ) ( Z116 A ) ( Z115 Z ) ;
- X403 ( Z112 A ) ( Z111 A ) ( Z110 A ) ( Z109 Z ) ;
- X401 ( Z106 A ) ( Z105 A ) ( Z104 A ) ( Z103 Z ) ;
- X315 ( Z60 A ) ( Z59 A ) ( Z58 A ) ( Z57 Z ) ;
- X313 ( Z54 A ) ( Z53 A ) ( Z52 A ) ( Z51 Z ) ;
- DIS051 ( Z216 A ) ( Z48 Z ) ;
- X311 ( Z48 A ) ( Z47 A ) ( Z46 A ) ( Z45 Z ) ;
- DIS041 ( Z215 A ) ( Z42 Z ) ; - X309 ( Z42 A ) ( Z41 A ) ( Z40 A )
    ( Z39 Z ) ;
- X307 ( Z35 A ) ( Z34 A ) ( Z33 A ) ( Z32 Z ) ;
- DIS031 ( Z214 A ) ( Z35 Z ) ; - DIS021 ( Z213 A ) ( Z29 Z ) ;
- X305 ( Z29 A ) ( Z28 A ) ( Z27 A ) ( Z26 Z ) ;
- DIS011 ( Z212 A ) ( Z23 Z ) ;
- X303 ( Z23 A ) ( Z22 A ) ( Z21 A ) ( Z20 Z ) ;
- DIS001 ( Z211 A ) ( Z17 Z ) ;
- X301 ( Z17 A ) ( Z16 A ) ( Z15 A ) ( Z14 Z ) ;
- X1000 ( E38A05 G ) ( E38A03 G ) ( E38A01 G ) ( E37 Z ) ;
- CNTEN ( Z38A05 Q ) ( E38A05 Q ) ( E25 A ) ;
- VIH20 ( E37 PI ) ( E25 PO ) ; - X0907 ( E36 B ) ( E35 B ) ( E25 Z ) ;
- CCLK0 ( F09 A ) ( E24 A ) ; - VIH19 ( E25 PI ) ( E24 PO ) ;
- X0906 ( E34 B ) ( E33 B ) ( E24 Z ) ; - CATH1 ( F09 Z ) ( E23 A ) ;
- VIH18 ( E24 PI ) ( E23 PO ) ; - CRLIN ( F08 Z ) ( E22 A ) ;
- VIH17 ( E23 PI ) ( E22 PO ) ; - X0904 ( E32 B ) ( E31 B ) ( E22 Z ) ;
- NXLIN ( F07 Z ) ( E21 A ) ; - VIH16 ( E22 PI ) ( E21 PO ) ;
- X0903 ( E30 B ) ( E29 B ) ( E21 Z ) ; - RPT1 ( F06 Z ) ( E20 A ) ;
- VIH15 ( E21 PI ) ( E20 PO ) ; - X0902 ( E28 B ) ( E27 B ) ( E20 Z ) ;
- AGISL ( F04 Z ) ( E19 A ) ; - VIH14 ( E20 PI ) ( E19 PO ) ;
- X0900 ( E35 A ) ( E33 A ) ( E32 A ) ( E30 A ) ( E28 A ) ( E26 A )
    ( E19 Z ) ;
- TSTCN ( Z38A05 QN ) ( E38A05 QN ) ( E18 A ) ;
- VIH13 ( E19 PI ) ( E18 PO ) ; - BCLK1 ( F01 A ) ( E17 A ) ;
- VIH12 ( E18 PI ) ( E17 PO ) ; - CLR0 ( F01 Z ) ( EE16 A ) ;
- VIH11 ( E17 PI ) ( EE16 PO ) ; - BCLKX1 ( Z216 C ) ( E17 Z )
    ( E16 C ) ; - CLRX0 ( Z38A05 CD ) ( Z38A03 CD ) ( Z38A01 CD )
    ( Z215 C ) ( E38A05 CD ) ( E38A03 CD ) ( E38A01 CD ) ( EE16 Z )
    ( E15 C ) ; - E38QN4 ( E38A04 QN ) ( E10 B ) ;
- CAX131 ( E38A04 Q ) ( E10 A ) ; - E38QN3 ( E38A03 QN ) ( E09 B ) ;
- CAX121 ( E38A03 Q ) ( E09 A ) ; - E38QN2 ( E38A02 QN ) ( E08 B ) ;
- CAX111 ( E38A02 Q ) ( E08 A ) ; - E38QN1 ( E38A01 QN ) ( E07 B ) ;
- CAX101 ( E38A01 Q ) ( E07 A ) ;
- SDD111 ( Z38A05 D ) ( Z205 Z ) ( E38A05 D ) ( E05 Z ) ;
- SDD121 ( Z38A04 D ) ( Z204 Z ) ( E38A04 D ) ( E04 Z ) ;
- X0901 ( E36 A ) ( E34 A ) ( E31 A ) ( E29 A ) ( E27 A ) ( E26 Z )
    ( D93 A ) ;
- VIH21 ( Z192 A ) ( E37 PO ) ( D92 A ) ;
- STRDENB0 ( Z206 B ) ( Z202 B ) ( Z201 B ) ( Z189 B ) ( Z188 B )
    ( F12 A ) ( E06 B ) ( E02 B ) ( E01 B ) ( D89 B ) ( D88 B ) ;
- STRDENA0 ( Z202 A ) ( Z201 A ) ( Z183 B ) ( Z182 B ) ( F12 Z )
    ( F01 H ) ( E02 A ) ( E01 A ) ( D83 B ) ( D82 B ) ;
- DAB151 ( F12 H ) ( D48 Z ) ; - DAA151 ( F08 B ) ( D47 Z ) ;
- X0415 ( D49 A ) ( D48 A ) ( D47 A ) ( D46 Z ) ;
- SDD151 ( Z38A01 D ) ( Z201 Z ) ( E38A01 D ) ( E01 Z ) ( D45 EN ) ;
- X0414 ( Z146 A ) ( D46 A ) ( D45 ZI ) ; - D151 ( E14 C ) ( D45 IO ) ;
- DAB141 ( F12 G ) ( D42 Z ) ; - DAA141 ( F08 A ) ( D41 Z ) ;
- X0413 ( D43 A ) ( D42 A ) ( D41 A ) ( D40 Z ) ;
- SDD141 ( Z38A02 D ) ( Z202 Z ) ( E38A02 D ) ( E02 Z ) ( D39 EN ) ;
- VIH60 ( D45 PI ) ( D39 PO ) ; - X0412 ( Z140 A ) ( D40 A ) ( D39 ZI ) ;
- D141 ( E13 C ) ( D39 IO ) ; - SDI131 ( E16 B ) ( D37 Z ) ;
- DAB131 ( F12 F ) ( D36 Z ) ; - DAA131 ( F07 B ) ( D35 Z ) ;
- X0411 ( D37 A ) ( D36 A ) ( D35 A ) ( D34 Z ) ;
- VIH58 ( Z193 Z ) ( D93 Z ) ( D33 PI ) ;
- SDD131 ( Z38A03 D ) ( Z203 Z ) ( E38A03 D ) ( E03 Z ) ( D33 EN ) ;
- VIH59 ( D39 PI ) ( D33 PO ) ; - X0410 ( Z134 A ) ( D34 A ) ( D33 ZI ) ;
- D131 ( E12 C ) ( D33 IO ) ; - SDI101 ( E15 B ) ( D19 Z ) ; ...
- X0315 ( C60 A ) ( C59 A ) ( C58 A ) ( C57 Z ) ;
- SDD071 ( Z211 Z ) ( E11 Z ) ( C56 EN ) ;
- VIH53 ( Z190 Z ) ( D90 Z ) ( D02 PI ) ( C56 PO ) ;
- X0314 ( Z57 A ) ( C57 A ) ( C56 ZI ) ;
- D071 ( E08 C ) ( C56 IO ) ; - SDI061 ( E11 B ) ( C54 Z ) ;
- DAB061 ( F09 H ) ( C53 Z ) ; - DAA061 ( F04 A ) ( C52 Z ) ;
- X0313 ( C54 A ) ( C53 A ) ( C52 A ) ( C51 Z ) ;
- SDD061 ( Z212 Z ) ( E12 Z ) ( C50 EN ) ;
- VIH52 ( Z189 Z ) ( D89 Z ) ( C56 PI ) ( C50 PO ) ;
- X0312 ( Z51 A ) ( C51 A ) ( C50 ZI ) ;
- D061 ( E07 C ) ( C50 IO ) ; - SDI051 ( E16 A ) ( C48 Z ) ;
- DAB051 ( F09 G ) ( C47 Z ) ; - DAA051 ( F01 G ) ( C46 Z ) ;
- X0311 ( C48 A ) ( C47 A ) ( C46 A ) ( C45 Z ) ;
- SDD051 ( Z213 Z ) ( E13 Z ) ( C44 EN ) ;
- VIH51 ( Z188 Z ) ( D88 Z ) ( C50 PI ) ( C44 PO ) ;
- X0310 ( Z45 A ) ( C45 A ) ( C44 ZI ) ;
- D051 ( E06 C ) ( C44 IO ) ; - SDI041 ( E15 A ) ( C42 Z ) ;
- DAB041 ( F09 F ) ( C41 Z ) ; - DAA041 ( F01 F ) ( C40 Z ) ;
- X0309 ( C42 A ) ( C41 A ) ( C40 A ) ( C39 Z ) ;
- SDD041 ( Z214 Z ) ( E14 Z ) ( C37 EN ) ;
- VIH50 ( Z187 Z ) ( D87 Z ) ( C44 PI ) ( C37 PO ) ;
- X0308 ( Z39 A ) ( C39 A ) ( C37 ZI ) ;
- D041 ( E05 C ) ( C37 IO ) ; - SDI031 ( E14 A ) ( C35 Z ) ;
- DAB031 ( F09 E ) ( C34 Z ) ; - DAA031 ( F01 E ) ( C33 Z ) ;
- X0307 ( C35 A ) ( C34 A ) ( C33 A ) ( C32 Z ) ;
- SDD031 ( Z215 Z ) ( E15 Z ) ( C31 EN ) ;
- VIH49 ( Z186 Z ) ( D86 Z ) ( C37 PI ) ( C31 PO ) ;
- X0306 ( Z32 A ) ( C32 A ) ( C31 ZI ) ;
- D031 ( E04 C ) ( C31 IO ) ; - SDI021 ( E13 A ) ( C29 Z ) ;
- DAB021 ( F09 D ) ( C28 Z ) ; - DAA021 ( F01 D ) ( C27 Z ) ;
- X0305 ( C29 A ) ( C28 A ) ( C27 A ) ( C26 Z ) ;
- SDD021 ( Z216 Z ) ( E16 Z ) ( C25 EN ) ;
- VIH48 ( Z185 Z ) ( D85 Z ) ( C31 PI ) ( C25 PO ) ;
- X0304 ( Z26 A ) ( C26 A ) ( C25 ZI ) ;
- D021 ( E03 C ) ( C25 IO ) ; - SDI011 ( E12 A ) ( C23 Z ) ;
- DAB011 ( F09 C ) ( C22 Z ) ; - DAA011 ( F01 C ) ( C21 Z ) ;
- X0303 ( C23 A ) ( C22 A ) ( C21 A ) ( C20 Z ) ;
- SDD011 ( Z209 Z ) ( E09 Z ) ( C19 EN ) ;
- VIH47 ( Z184 Z ) ( D84 Z ) ( C25 PI ) ( C19 PO ) ;
- X0302 ( Z20 A ) ( C20 A ) ( C19 ZI ) ;
- D011 ( E02 C ) ( C19 IO ) ; - SDI001 ( E11 A ) ( C17 Z ) ;
- DAB001 ( F09 B ) ( C16 Z ) ; - DAA001 ( F01 B ) ( C15 Z ) ;
- X0301 ( Z14 A ) ( C17 A ) ( C16 A ) ( C15 A ) ( C14 Z ) ;
- VIH45 ( Z182 Z ) ( D82 Z ) ( C13 PI ) ;
- SDD001 ( Z210 Z ) ( E10 Z ) ( C13 EN ) ;
- VIH46 ( Z183 Z ) ( D83 Z ) ( C19 PI ) ( C13 PO ) ;
- X0300 ( C14 A ) ( C13 ZI ) ; - D001 ( E01 C ) ( C13 IO ) ;
- CCLKB0 ( Z234 Z ) ( Z189 A ) ( E34 Z ) ( D89 A ) ( C11 A ) ;
- VIH10 ( EE16 PI ) ( C11 PO ) ;
- STRAAA ( Z206 A ) ( E06 A ) ( C11 Z ) ;
- CCLKA0 ( Z233 Z ) ( Z188 A ) ( E33 Z ) ( D88 A ) ( C10 A ) ;
- VIH9 ( C11 PI ) ( C10 PO ) ;
- STRB00 ( Z192 B ) ( D92 B ) ( C10 Z ) ;
- CRLINB1 ( Z232 Z ) ( Z187 A ) ( E32 Z ) ( D87 A ) ( C09 A ) ;
- VIH8 ( C10 PI ) ( C09 PO ) ;
- STRA00 ( Z187 B ) ( D87 B ) ( C09 Z ) ;
- CRLINA1 ( Z231 Z ) ( Z186 A ) ( E31 Z ) ( D86 A ) ( C08 A ) ;
- VIH7 ( C09 PI ) ( C08 PO ) ;
- X10001 ( E38A04 G ) ( E38A02 G ) ( C08 Z ) ;
- NXLINB1 ( Z230 Z ) ( Z185 A ) ( E30 Z ) ( D85 A ) ( C07 A ) ;
- VIH6 ( C08 PI ) ( C07 PO ) ;
- CLRX00 ( Z38A04 CD ) ( Z38A02 CD ) ( E38A04 CD ) ( E38A02 CD )
    ( C07 Z ) ;
- NXLINA1 ( Z229 Z ) ( Z184 A ) ( E29 Z ) ( D84 A ) ( C06 A ) ;
- VIH5 ( C07 PI ) ( C06 PO ) ;
- STRBB0 ( Z205 B ) ( Z193 B ) ( E05 B ) ( D93 B ) ( C06 Z ) ;
- RPTB1 ( Z228 Z ) ( Z183 A ) ( E28 Z ) ( D83 A ) ( C05 A ) ;
- VIH4 ( C06 PI ) ( C05 PO ) ;
- STRAA0 ( Z205 A ) ( Z186 B ) ( E05 A ) ( D86 B ) ( C05 Z ) ;
- RPTA1 ( Z227 Z ) ( Z182 A ) ( E27 Z ) ( D82 A ) ( C04 A ) ;
- VIH3 ( C05 PI ) ( C04 PO ) ;
- STRB0 ( Z204 B ) ( Z203 B ) ( Z191 B ) ( Z190 B ) ( E04 B )
```

```
    ( E03 B ) ( D91 B ) ( D90 B ) ( C04 Z ) ;
- CNTENB0 ( Z236 Z ) ( Z191 A ) ( E36 Z ) ( D91 A ) ( C02 A ) ;
- VIH2 ( C04 PI ) ( C02 PO ) ;
- STRA0 ( Z204 A ) ( Z203 A ) ( Z185 B ) ( Z184 B ) ( E04 A )
    ( E03 A ) ( D85 B ) ( D84 B ) ( C02 Z ) ;
- CNTENA0 ( Z235 Z ) ( Z190 A ) ( E35 Z ) ( D90 A ) ( C01 A ) ;
- VIH1 ( C02 PI ) ( C01 PO ) ; - CALCH ( E37 A ) ( C01 Z ) ;
#
```

Scan Chain Synthesis Example

You define the scan chain in the `COMPONENTS` and `SCANCHAINS` sections in your DEF file.

```
COMPONENTS 100 ;
- SIN MUX ;
- SOUT PAD ;
- C1 SDFF ;
- C2 SDFF ;
- C3 SDFF ;
- C4 SDFF ;
- B1 BUF ;
- A1 AND ; ...
END COMPONENTS

NETS 150 ;
- N1 (C1 SO) (C3 SI) ;
- N2 (C3 SO) (A1 A) ; ...
END NETS
```

You do not need to define any scan nets in the `NETS` section. This portion of the `NETS` section shows the effect of the scan chain process on existing nets that use components you specify in the `SCANCHAINS` section.

```
SCANCHAINS 1 ;
- SC
    + COMMONSCANPINS (IN SI) (OUT SO)
    + START SIN Z2
    + FLOATING C1 C2 C3
    + ORDERED C4 B1 (IN A) (OUT Q) ;
    + STOP SOUT A ;
END SCANCHAINS
```

Because components C1, C2, and C3 are floating, `TROUTE SCANCHAIN` can synthesize them in any order in the chain. `TROUTE` synthesizes ordered components (C4 and B1) in the order you specify.

B

Optimizing LEF Technology for Place and Route

This appendix contains the following information.

Overview

This appendix provides guidelines for defining the optimized technology section in the LEF file to get the best performance using Cadence® place-and-route tools, especially Cadence Ultra Router. The LEF syntax shown is based on Silicon Ensemble® Place-and-Route 5.2 or newer.

For the following guidelines, the preferred routing direction for `metal1` and all other odd metal layers is horizontal. The preferred routing direction for `metal2` and all other even metal layers is vertical. Standard cells are arranged in horizontal rows.

This appendix discusses the following LEF statements.

```
NOWIREEXTENSIONATPIN { ON | OFF }


LAYER layerName
     TYPE ROUTING ;
     PITCH distance ;
     WIDTH defWidth ;
     SPACING minSpacing [RANGE minwidth maxwidth] ;
     WIREEXTENSION value ;

END layerName

VIA viaName DEFAULT
     [TOPSTACKONLY]
     LAYER layerName RECT pt pt ; ...

END viaName

SPACING
     SAMENET
        layerName layerName minSpace [STACK] ;

END SPACING
```

Guidelines for No Wire Extension at Pin

By default, wires extend one half of the default routing width at pins. To prevent this wire extension at pins when the pin width is narrower than the wire width, use the following statement in the LEF file.

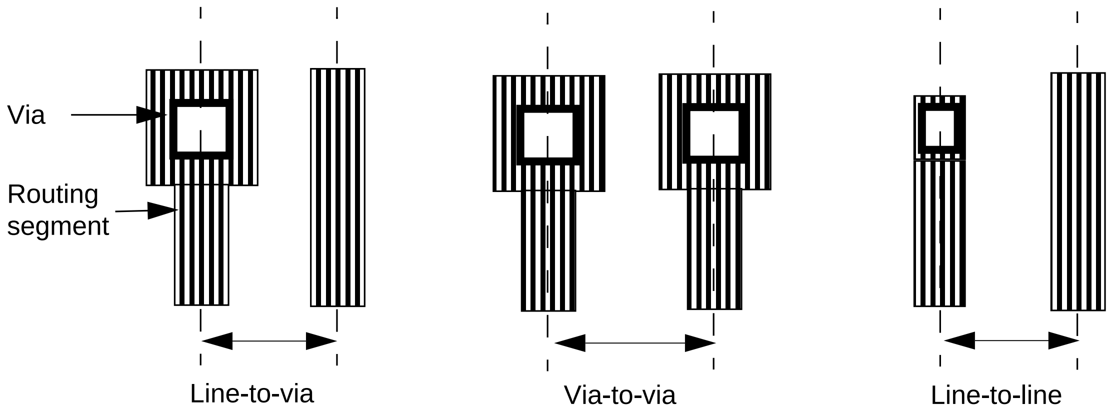
```
NOWIREEXTENSIONATPIN ON ;
```

To control the wire extension at vias, double-check the via size and metal enclosure rule. Then use the `WIREEXTENSION` statement in the appropriate LAYER section.

Guidelines for Routing Pitch

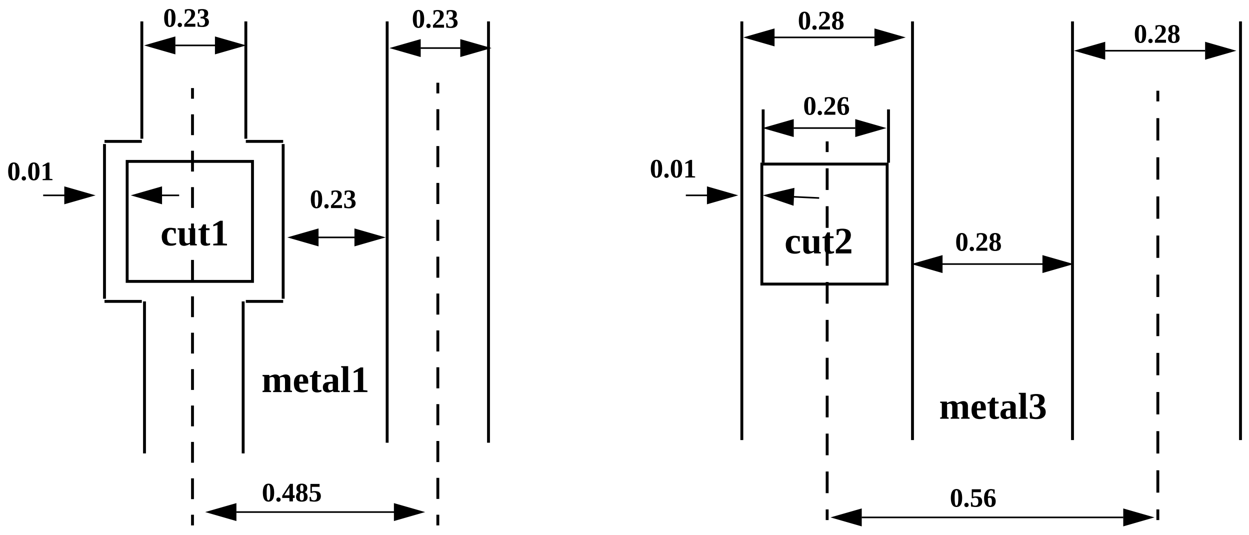
The following is a summary for choosing the right pitch for an existing design library. For detailed information on determining routing pitch, refer to the *Cadence Abstract Generator User Guide*.

Pitch Measurement



DESIGN RULE No. 1

W.1	Minimum width of `metal1` = 0.23 um
S.1	Minimum space between two `metal1` regions = 0.23 um
W.2	Minimum and maximum width of `cut1` = 0.26 um
E.1	Minimum extension of `metal1` beyond `cut1` = 0.01 um
W.3	Minimum width of `metal3` = 0.28 um
S.2	Minimum space between two `metal3` regions = 0.28 um
W.4	Minimum and maximum width of `cut2` = 0.26 um
E.2	Minimum extension of `metal1` beyond `cut2` = 0.01 um



Although the minimum `metal1` routing pitch is 0.485um from the design rule, you should use 0.56um instead, to match the `metal3` routing pitch in the same preferred direction.

LEF Construct No. 1

```
LAYER metal1
     TYPE ROUTING ;
     WIDTH 0.23 ;
     SPACING 0.23 ;
     PITCH 0.56 ;
     DIRECTION HORIZONTAL ;
END metal1


LAYER metal3
     TYPE ROUTING ;
     WIDTH 0.28 ;
     SPACING 0.28 ;
     PITCH 0.56 ;
     DIRECTION HORIZONTAL ;
END metal3
```

Recommendations

- Use line-to-via spacing for both the horizontal and vertical direction.
- Allow diagonal vias with the routing pitch.
- Align the routing pitch for `metal1` and `metal2`, with the pins inside the standard cells.
- Have uniform routing pitch in the same preferred direction. The pitch ratio should be 2 - 3 or 1 - 2. It is better to define the `metal1` pitch larger than necessary in order to achieve a 1 - 1 ratio because the `metal1` width is usually smaller the `metal2` and `metal3` widths.

Pitch Recommendations for Library Development

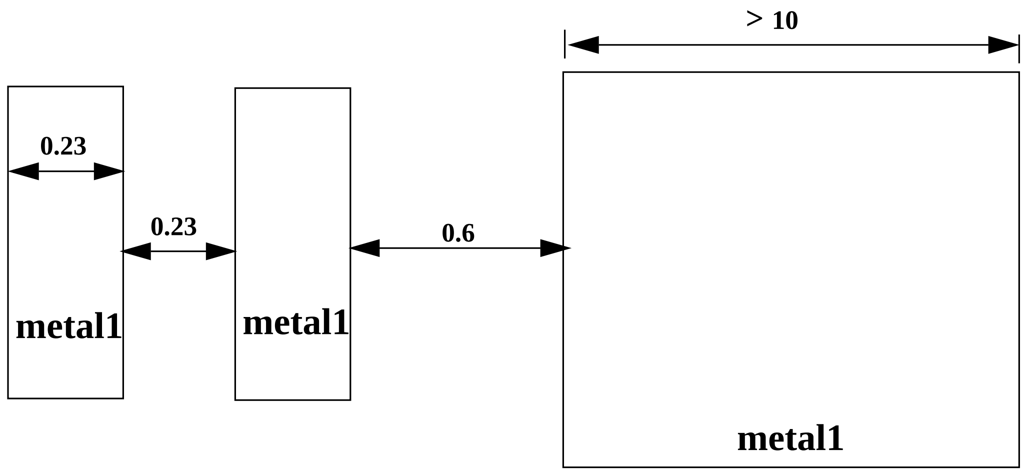
- All pins should be on the grid, and only those portions of the pins that are accessible to the router should be modeled as pins. For example, 45 degree pin geometry.
- The height of the cell should be the even multiple of the `metal1` pitch, and the width of the cell should be the even multiple of the `metal2` pitch.
- The blockage modeling, especially for `metal1`, should be simplified as much as possible. For example, it is very common for the entire area within the cell boundary to be obstructed in `metal1`, so use a single rectangular blockage instead of many small blockages.

Guidelines for Wide Metal Spacing

The `SPACING` statement in the LEF `LAYER` section is applied to both regular and special wires. You can use the Cadence® ultra router option `frouteUseRangeRule` to determine which objects to check against the `SPACING RANGE` statement. The default checks both pin and obstruction.

DESIGN RULE No. 2

S.1	Minimum space between two `metal1` regions = 0.23 um
S.2	Minimum space between metal lines with one or both metal line width and length are greater than 10um = 0.6 um



LEF CONSTRUCT No. 2

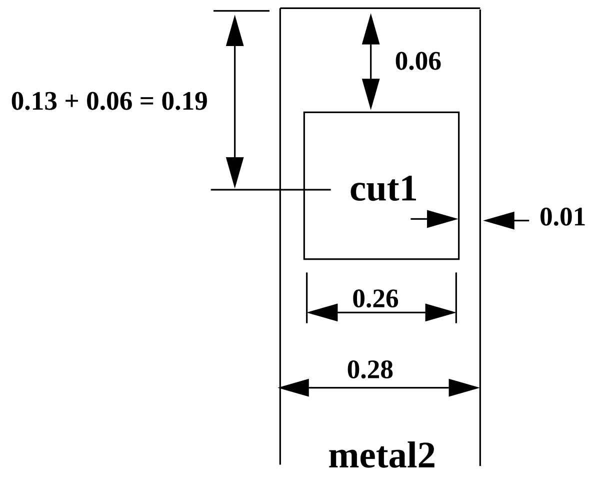
```
LAYER metal1
     WIDTH 0.23 ;
     SPACING 0.23 ;
     SPACING 0.6 RANGE 10.002 1000 ;

END metal1
```

Guidelines for Wire Extension at Vias

The following guidelines are for wire extension at vias.

DESIGN RULE No. 3

W.1	Minimum and maximum width of `cut1` = 0.26 um
W.2	Minimum width of `metal2` = 0.28 um
E.1	Minimum extension of `metal2` beyond `cut1` = 0.01 um
E.2	Minimum extension of `metal2` end-of-line region beyond `cut1` = 0.06 um



LEF CONSTRUCT No. 3

```
LAYER metal2
     TYPE ROUTING ;
     WIDTH 0.28 ;
     SPACING 0.28 ;
     PITCH 0.56 ;
     WIREEXTENSION 0.19 ;
     DIRECTION VERTICAL ;
END metal2



VIA via23 DEFAULT
     LAYER metal2 ;
          RECT -0.14 -0.14 0.14 0.14 ;        # Use square via
     LAYER cut2 ;
```

```
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal3 ;
          RECT -0.14 -0.14 0.14 0.14 ;        # Use square via
END via23
```

Recommendations

- Use the `WIREEXTENSION` statement instead of defining multiple vias because the width of the `metal2` in `cut1` is the same as the default routing width of the `metal2` layer.
- Define the `DEFAULT VIA` as a square via.

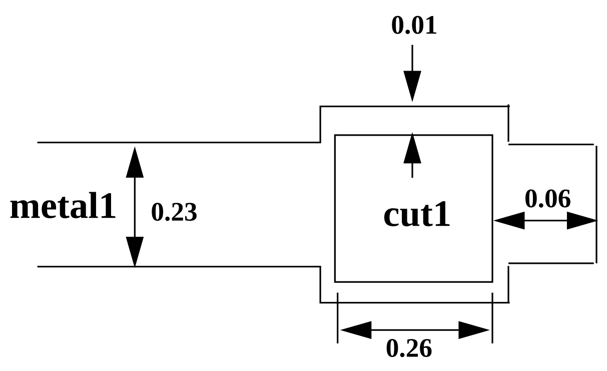
Guidelines for Default Vias

The following guidelines are for default vias.

DESIGN RULE No. 4

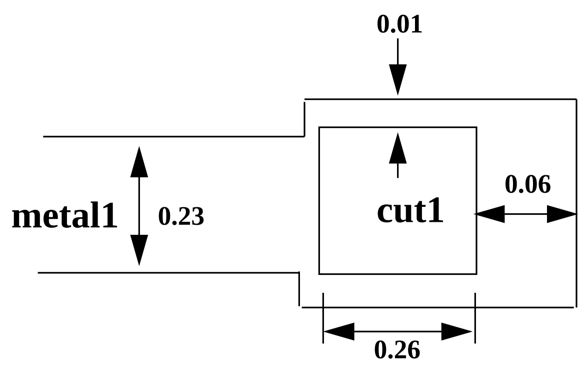
W.1	Minimum width of `metal1` = 0.23 um
W.2	Minimum and maximum width of `cut1` = 0.26 um
E.1	Minimum extension of `metal1` beyond `cut1` = 0.01 um
E.2	Minimum extension of `metal1` end-of-line region beyond `cut1` = 0.06 um

Case A:



Use `WIREEXTENSION` and
square `DEFAULT VIA`

Case B:



Use Horizontal and Vertical `DEFAULT VIA`s

LEF CONSTRUCT No. 4 (Case B)

```
LAYER metal1
     TYPE ROUTING ;
     WIDTH 0.23 ;
     SPACING 0.23 ;
     PITCH 0.56 ;
     DIRECTION HORIZONTAL ;
END metal1


VIA via12_H DEFAULT
     LAYER metal1 ;
          RECT -0.19 -0.14 0.19 0.14 ;        # metal1 end-of-line
          extension 0.6 in both directions
     LAYER cut1 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal2 ;
          RECT -0.14 -0.14 0.14 0.14 ;

END via12_H


VIA via12_V DEFAULT
     LAYER metal1 ;
          RECT -0.14 -0.19 0.14 0.19 ;        # metal1 end-of-line
          extension 0.6 in both directions
     LAYER cut1 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal2 ;
          RECT -0.14 -0.14 0.14 0.14 ;

END via12_V
```

Recommendations

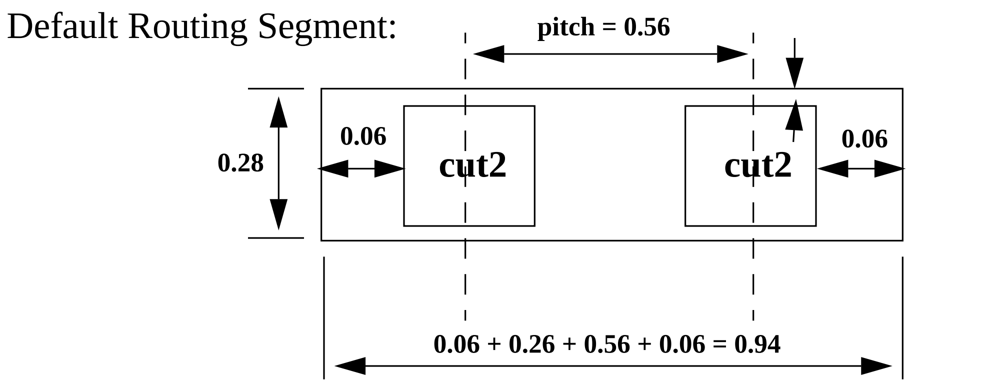
- If the width of the end-of-line metal extension is the same as the default metal routing width, as in Case A, use the `WIREEXTENSION` statement in the LEF `LAYER` section, and define a square via in the `DEFAULT VIA` section.
- If the width of the end-of-line metal extension is the same as the width of the via metal, as in Case B, define one horizontal `DEFAULT VIA` and one vertical `DEFAULT VIA` to cover the required metal extension area in both pregerred and non-preferred routing directions. Do not use the `WIREEXTENSION` statement in the LEF `LAYER` section.

Guidelines for Stack Vias (MAR Vias) and Samenet Spacing

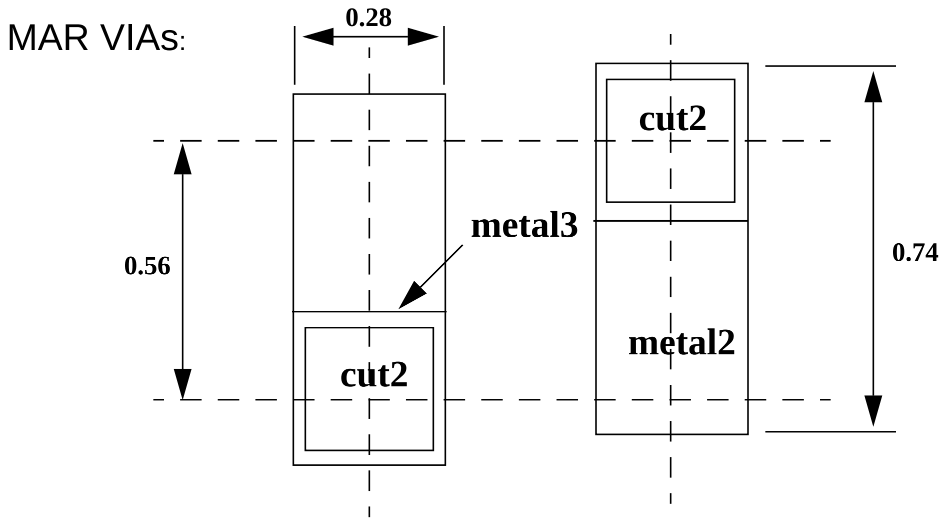
The following guidelines are for stack vias (minimum area rule) and `SAMENET SPACING`.

DESIGN RULE No. 5

W.1	Minimum width of `metal2` = 0.28 um
W.2	Minimum and maximum width of `cut2` = 0.26 um
E.1	Minimum extension of `metal2` beyond `cut2` = 0.01 um
A.1	Minimum area of `metal2` = 0.2025 um
C.1	`Cut2` can be fully or partially stacked on `cut1`, contact or any combination
W.1	Minimum width of `metal3` = 0.28 um
W.2	Minimum and maximum width of `cut3` = 0.26 um
E.1	Minimum extension of `metal2` beyond `cut3` = 0.01 um
A.1	Minimum area of `metal3` = 0.2025 um
C.1	`Cut3` can be fully or partially stacked on `cut2`, `cut1`, contact or any combination



Minimum routing area of metal3 = 0.28 x 0.94 = 0.2632 > 0.2.25 (MAR)



via23_stack_north via23_stack_south

LEF CONSTRUCT No. 5

```
VIA via23_stack_north DEFAULT TOPOFSTACKONLY
     LAYER metal2 ;
          RECT -0.14 -0.14 0.14 0.6 ;   # MAR = 0.28 x 0.74
     LAYER cut2 ;
```

```
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal3 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via23_stack_north


VIA via23_stack_south DEFAULT TOPOFSTACKONLY
     LAYER metal2 ;
          RECT -0.14 -0.6 0.14 0.14 ;   # MAR = 0.28 x 0.74
     LAYER cut2 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal3 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via23_stack_south


VIA via34_stack_east DEFAULT TOPOFSTACKONLY
     LAYER metal3 ;
          RECT -0.14 -0.14 0.6 0.14 ;   # MAR = 0.28 x 0.74
     LAYER cut3 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal4 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via34_stack_east


VIA via34_stack_west DEFAULT TOPOFSTACKONLY
     LAYER metal3 ;
          RECT -0.6 -0.14 0.14 0.14 ;   # MAR = 0.28 x 0.74
     LAYER cut3 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal4 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via34_stack_west


SPACING
     SAMENET metal2 metal2 0.28  STACK ;
     SAMENET metal3 metal3 0.28  STACK ;

     SAMENET  cut2  cut2  0.26 ;
     SAMENET  cut3  cut3  0.26 ;

     SAMENET  cut1  cut2  0.0 STACK ;
     SAMENET  cut2  cut3  0.0 STACK ;
END SPACING
```

Recommendations

- The minimum metal routing segment (two vias between one pitch grid) with or without end-of-line metal extension should automatically satisfy the minimum area rule.
- If vias are stackable, create the `TOPSTACKONLY` vias with a rectangular shape blocking only one neighboring grid for both sides of the preferred routing direction. In other words, one north oriented and one south oriented for vertical-preferred routing layers, and one east oriented and one west oriented for horizontal-preferred routing layers.
- Use slightly larger dimensions for the via size to make them an even number, so they snap to the manufacturing grids.
- The `STACK` keyword in the `SAMENETSPACING` statements only allows vias to be fully overlapped (stacked) by SROUTE commands. To allow vias to be partially overlapped, set the environment variable `SROUTE.ALLOWOVERLAPINSTACKVIA` to `TRUE`.
- The `metal1` layer does not require a MAR via because all metal1 pins should satisfy the minimum area rules.

Example of an Optimized LEF Technology File

```
VERSION 5.2 ;

NAMESCASESENSITIVE ON ;

BUSBITCHARS "[]" ;


UNITS
     DATABASE MICRONS 100  ;

END UNITS



LAYER metal1
     TYPE ROUTING ;
     WIDTH 0.23 ;
     SPACING 0.23 ;
     SPACING 0.6 RANGE 10.02 1000 ;
     PITCH 0.56 ;
     DIRECTION HORIZONTAL ;

END metal1



LAYER cut1
     TYPE CUT ;

END cut1
```

```
LAYER metal2
     TYPE ROUTING ;
     WIDTH 0.28 ;
     SPACING 0.28 ;
     SPACING 0.6 RANGE 10.02 1000 ;
     PITCH 0.56 ;
     WIREEXTENSION 0.19 ;
     DIRECTION VERTICAL ;
END metal2


LAYER cut2
     TYPE CUT ;
END cut2


LAYER metal3
     TYPE ROUTING ;
     WIDTH 0.28 ;
     SPACING 0.28 ;
     SPACING 0.6 RANGE 10.02 1000 ;
     PITCH 0.56 ;
     WIREEXTENSION 0.19 ;
     DIRECTION HORIZONTAL ;
END metal3


LAYER cut3
     TYPE CUT ;
END cut3


LAYER metal4
     TYPE ROUTING ;
     WIDTH 0.28 ;
     SPACING 0.28 ;
     SPACING 0.6 RANGE 10.02 1000 ;
     PITCH 0.56 ;
     WIREEXTENSION 0.19 ;
     DIRECTION VERTICAL ;
END metal4
```

```
LAYER cut4
     TYPE CUT ;
END cut4


LAYER metal5
     TYPE ROUTING ;
     WIDTH 0.28 ;
     SPACING 0.28 ;
     SPACING 0.6 RANGE 10.02 1000 ;
     PITCH 0.56 ;
     WIREEXTENSION 0.19 ;
     DIRECTION HORIZONTAL ;
END metal5


LAYER cut5
     TYPE CUT ;
END cut5


LAYER metal6
     TYPE ROUTING ;
     WIDTH 0.44 ;
     SPACING 0.46 ;
     SPACING 0.6 RANGE 10.02 1000 ;
     PITCH 1.12 ;
     DIRECTION VERTICAL ;
END metal6
```

### start DEFAULT VIA ###

```
VIA via12_H DEFAULT
     LAYER metal1 ;
          RECT -0.19 -0.14 0.19 0.14 ;     # metal1 end-of-line ext 0.6
     LAYER cut1 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal2 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via12_H


VIA via12_V DEFAULT
     LAYER metal1 ;
          RECT -0.14 -0.19 0.14 0.19 ;     # metal1 end-of-line ext 0.6
     LAYER cut1 ;
```

```
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal2 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via12_V


VIA via23 DEFAULT
     LAYER metal2 ;
          RECT -0.14 -0.14 0.14 0.14 ;
     LAYER cut2 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal3 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via23


VIA via34 DEFAULT
     LAYER metal3 ;
          RECT -0.14 -0.14 0.14 0.14 ;
     LAYER cut3 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal4 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via34


VIA via45 DEFAULT
     LAYER metal4 ;
          RECT -0.14 -0.14 0.14 0.14 ;
     LAYER cut4 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal5 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via45


VIA via56_H DEFAULT
     LAYER metal5 ;
          RECT -0.24 -0.19 0.24 0.19 ;
     LAYER cut5 ;
          RECT -0.18 -0.18 0.18 0.18 ;
     LAYER metal6 ;
          RECT -0.27 -0.27 0.27 0.27 ;
END via56_H
```

```
VIA via56_V DEFAULT
     LAYER metal5 ;
          RECT -0.19 -0.24 0.19 0.24 ;
     LAYER cut5 ;
          RECT -0.18 -0.18 0.18 0.18 ;
     LAYER metal6 ;
          RECT -0.27 -0.27 0.27 0.27 ;
END via56_V
```

### end DEFAULT VIA ###

### start STACK VIA ###

```
VIA via23_stack_north DEFAULT TOPOFSTACKONLY
     LAYER metal2 ;
          RECT -0.14 -0.14 0.14 0.6 ;   # MAR = 0.28 x 0.74
     LAYER cut2 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal3 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via23_stack_north


VIA via23_stack_south DEFAULT TOPOFSTACKONLY
     LAYER metal2 ;
          RECT -0.14 -0.6 0.14 0.14 ;   # MAR = 0.28 x 0.74
     LAYER cut2 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal3 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via23_stack_south


VIA via34_stack_east DEFAULT TOPOFSTACKONLY
     LAYER metal3 ;
          RECT -0.14 -0.14 0.6 0.14 ;   # MAR = 0.28 x 0.74
     LAYER cut3 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal4 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via34_stack_east


VIA via34_stack_west DEFAULT TOPOFSTACKONLY
     LAYER metal3 ;
          RECT -0.6 -0.14 0.14 0.14 ;   # MAR = 0.28 x 0.74
     LAYER cut3 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal4 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via34_stack_west


VIA via45_stack_north DEFAULT TOPOFSTACKONLY
     LAYER metal4 ;
          RECT -0.14 -0.14 0.14 0.6 ;   # MAR = 0.28 x 0.74
     LAYER cut4 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal5 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via45_stack_north


VIA via45_stack_south DEFAULT TOPOFSTACKONLY
     LAYER metal4 ;
          RECT -0.14 -0.6 0.14 0.14 ;   # MAR = 0.28 x 0.74
     LAYER cut4 ;
          RECT -0.13 -0.13 0.13 0.13 ;
     LAYER metal5 ;
          RECT -0.14 -0.14 0.14 0.14 ;
END via45_stack_south


VIA via56_stack_east DEFAULT TOPOFSTACKONLY
     LAYER metal5 ;
          RECT -0.19 -0.19 0.35 0.19 ;   # MAR = 0.38 x 0.54
     LAYER cut5 ;
          RECT -0.18 -0.18 0.18 0.18 ;
     LAYER metal6 ;
          RECT -0.27 -0.27 0.27 0.27 ;
END via56_stack_east


VIA via56_stack_west DEFAULT TOPOFSTACKONLY
     LAYER metal5 ;
          RECT -0.35 -0.19 0.19 0.19 ;   # MAR = 0.38 x 0.54
     LAYER cut5 ;
          RECT -0.18 -0.18 0.18 0.18 ;
     LAYER metal6 ;
          RECT -0.27 -0.27 0.27 0.27 ;
END via56_stack_west
```

### end STACK VIA ###

```
SPACING
     SAMENET metal1 metal1 0.23  STACK ;
     SAMENET metal2 metal2 0.28  STACK ;
     SAMENET metal3 metal3 0.28  STACK ;
     SAMENET metal4 metal4 0.28  STACK ;
     SAMENET metal5 metal5 0.28  STACK ;
     SAMENET metal6 metal6 0.46  STACK ;

     SAMENET  cut1  cut1  0.26 ;
     SAMENET  cut2  cut2  0.26 ;
     SAMENET  cut3  cut3  0.26 ;
     SAMENET  cut4  cut4  0.26 ;
     SAMENET  cut5  cut5  0.35 ;

     SAMENET  cut1  cut2  0.0 STACK ;
     SAMENET  cut2  cut3  0.0 STACK ;
     SAMENET  cut3  cut4  0.0 STACK ;
     SAMENET  cut4  cut5  0.0 STACK ;

END SPACING
```

C

Calculating and Fixing Process Antenna Violations

This appendix describes process antenna violations and how you can use the router to correct them. It includes the following sections:

Overview

During deep submicron wafer fabrication, gate damage can occur when excessive static charges accumulate and discharge, passing current through a gate. If the area of the layer connected directly to the gate or connected to the gate through lower layers is large relative to the area of the gate and the static charges are discharged through the gate, the discharge can damage the oxide that insulates the gate and cause the chip to fail. This phenomenon is called the *process antenna effect* (PAE).

To determine the extent of the PAE, the router calculates the area of the layer relative to the area of the gates connected to it, or connected to it through lower layers. The number it calculates is called the *antenna ratio*. Each foundry sets a maximum allowable antenna ratio for the chips it fabricates.

For example, assume a foundry sets a maximum allowable antenna ratio of 500. If a net has two input gates that each have an area of 1 square micron, any metal layers that connect to the gates and have an area larger than 1,000 square microns have process antenna violations because they would cause the antenna ratio to be higher than 500:

$$\text{Antenna Ratio} = \frac{\text{Area of metal layer}}{\text{Area of gates}} \qquad 500 = \frac{1000}{1 + 1}$$

To tell the router the values to use when it calculates the antenna ratio, you set antenna keywords in the LEF and DEF files. The router measures potential damage caused by PAE by checking the ratio it calculates against the values specified by the antenna keywords. When it finds a net whose antenna ratio for a specified layer exceeds the maximum allowed value for that layer, it finds a *process antenna violation* and attempts to fix it using one or both of the following methods:

- Changing the routing so the routing layers connected to a gate or connected to a gate through lower layers are not so large that they build enough static charge to damage the gate
- Inserting diodes that protect the gate by providing an alternate path to discharge the static charge

LEF can specify several types of antenna ratios, including ratios for PAE damage on one layer only and ratios calculated by adding accumulated damage on several layers. In addition, LEF can specify ratios based on the area of the metal wires or the cut area of vias.

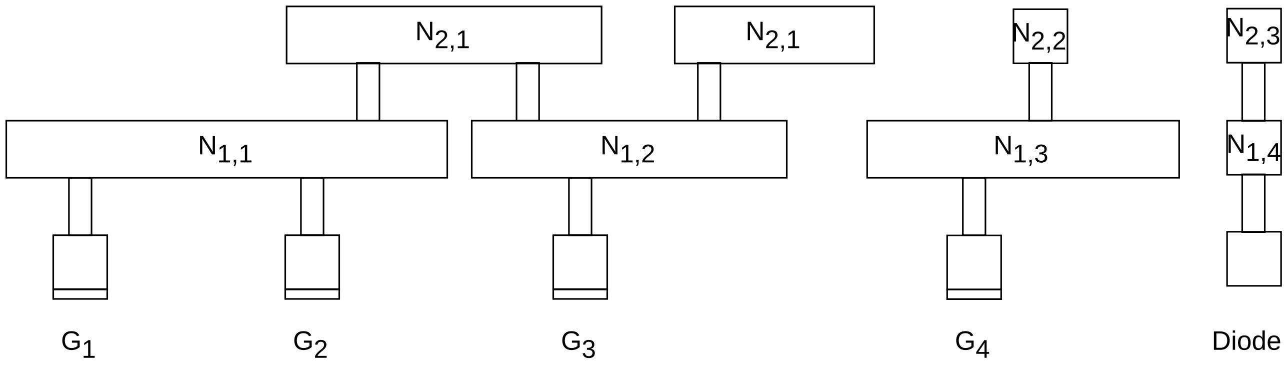
What Are Process Antennas?

In a chip manufacturing process, metal layers are built up, layer by layer, starting with the first-level metal layer (usually referred to as *metal1*). Next, the *metal1-metal2* vias are created, then the second-level metal layer, then *metal2-metal3* vias, and so on.

On each metal layer, metal is initially deposited so it covers the entire chip. Then, the unneeded portions of the metal are removed by etching, typically in plasma (charged particles).

Figure C-1 on page 255 shows a section of an imaginary chip after the unneeded metal from *metal2* is removed.

Figure C-1



In the figure,

- Gate areas for transistors are labelled G_k, where *k* is a sequential number starting with 1.
- Wire segments are labelled $N_{i,j}$
  - N signifies that the wire segment is an electrically connected node
  - *i* specifies the metal layer to which the node belongs
  - *j* is a sequential number for the node on that metal layer
- Nodes are labelled so that all pieces of the metal geometry on layer *metal*$_i$ that are electrically connected by conductors at layers below *metal*$_i$ belong to the same node. For example, the two *metal2* wire segments that belong to node $N_{2,1}$ are electrically connected to gates G_1, G_2, and G_3 by a piece of wire on *metal1* (labelled $N_{1,2}$).

Thick oxide insulates the already-fabricated structures below *metal2*, preventing them from direct contact with the plasma. The *metal2* geometries, however, are exposed to the plasma, and collect charge from it. As the metal geometries collect charge, they build up voltage potential.

Because the metal geometries collect charge during the metallization process, they are referred to as process antennas. In general, the more area covered by the metal geometries that are exposed to the plasma (that is, the larger the process antennas), the more charge they can collect.

In Figure C-1 on page 255, note the following:

- Node $N_{1,1}$ is electrically connected to gates G_1 and G_2.
- Node $N_{1,2}$ is electrically connected to gate G_3.
- Node $N_{2,1}$ (node $N_{2,1}$ has two pieces of metal) is electrically connected to gates G_1, G_2, and G_3.
- Node $N_{1,3}$ and node $N_{2,2}$ are electrically connected to gate G_4.
- Node $N_{1,4}$ and node $N_{2,3}$ are electrically connected to the diffusion (diode).

What Is the Process Antenna Effect (PAE)?

If the voltage potential across the gate oxide becomes large enough to cause current to flow across the gate oxide, from the process antennas to the gates to which the process antennas are electrically connected, the current can damage the gate oxide. The process antenna effect (PAE) is the term used to describe the build-up of charge and increase in voltage potential. The larger the total gate area that is electrically connected to the process antennas on a specific layer, the more charge the connected gates can withstand.

In the imaginary chip in Figure C-1 on page 255, if the current were to flow, the following would happen, as a result of the node-gate connections:

- The charge collected by process antennas on nodes $N_{1,1}$, $N_{1,2}$, and $N_{2,1}$ would be discharged through one or more of gates G_1, G_2, and G_3.
- The charge collected by process antennas on nodes $N_{1,3}$ and $N_{2,2}$ would be discharged through gate G_4.
- The charge collected by process antennas on node $N_{1,4}$ and $N_{2,3}$ would be discharged through the diode.

What Is the Antenna Ratio?

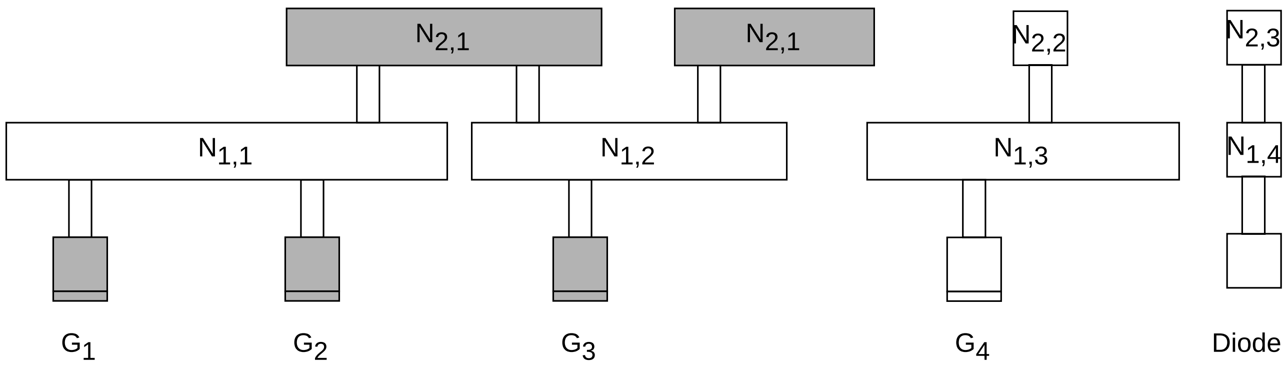
Because the total gate area that is electrically connected to a node (and therefore connected to the process antennas) determines the amount of charge from the process antennas the electrically connected gates can withstand, and because the size of the process antennas connected to the node determines how much charge the antennas collect, it is useful to calculate the ratio of the size of the process antennas on a node to the size of the gate area that is electrically connected to the node. This is the antenna ratio. The greater the antenna ratio, the greater the potential for damage to the gate oxide.

If you check a chip and obtain an antenna ratio greater than the threshold specified by the foundry, gate damage is likely to occur.

Figure C-2 on page 257 shows the same section of the imaginary chip as the previous figure. The shaded areas in this figure represent the process antennas on node $N_{2,1}$ and the gates to which they connect: gates G_1, G_2, and G_3. The shaded gates discharge the electricity collected by the process antennas on node $N_{2,1}$.

Figure C-2

$$\text{Antenna Ratio} = \frac{\text{Area of process antennas on a node}}{\text{Area of gates electrically connected to the node}}$$



What Can Be Done to Improve the Antenna Ratio?

If there is an alternate path for the current to flow, the charge on the node can be discharged through the alternate path before the voltage potential reaches a level that damages the gate. For example, a Zener diode, which allows current to flow in the reverse direction when the reverse bias reaches a specified breakdown voltage, provides an alternate path, and helps avoid building up so much charge at the node that the charge is discharged through the gate oxide. Diffusion features that form the output of a logic gate (source and drain of transistors) can provide such an alternate discharge path.

Routers typically use two methods to decrease the antenna ratio:

- Changing the routing by breaking the metal layers into smaller pieces
- Inserting antenna diode cells to discharge the current

Both of these methods supply alternate paths for the current. For details about how to specify antenna diode cells, see "Using Antenna Diode Cells" on page 279.

Using Process Antenna Keywords in the LEF and DEF Files

You tell the router the values to use for the gate, diffusion, and metal areas by setting values for process antenna keywords in the LEF and DEF files for your design. You also tell the router the values to use for the threshold process antenna ratios by setting the keywords.

The following table lists LEF version 5.5 antenna keywords.

If the keyword ends with ...	It refers to ...	Examples
`area`	Area of the gates or diffusion Measured in square microns	`ANTENNADIFFAREA` `ANTENNAGATEAREA`

`factor`	Area multiplier used for the metal nodes	`ANTENNAAREAFACTOR`
		`ANTENNASIDEAREAFACTOR`
		
		Note: Use `DIFFUSEONLY` if you want the multiplier to apply only when connecting to diffusion. For more information, see "Using DiffUseOnly" on page 280.

If the keyword ends with ...	It refers to ...	Examples

`ratio`	Relationship the router is calculating
	
	`Cum` is used in keywords for cumulative antenna ratio.
		`ANTENNAAREARATIO`
		`ANTENNASIDEAREARATIO`
		`ANTENNADIFFAREARATIO`
		`ANTENNADIFFSIDEAREARATIO`
		`ANTENNACUMAREARATIO`
		`ANTENNACUMSIDEAREARATIO`
		`ANTENNACUMDIFFAREARATIO`
		`ANTENNACUMDIFFSIDEAREARATIO`

Calculating Antenna Ratios

Tools should calculate antenna ratios using one of the following models:

- The partial checking model

  Using this model, you calculate damage to gates by process antennas on one layer. For example, if you use the partial checking model to calculate the PAE referred to a gate from *metal3*, you do not consider any potential damages referred to that gate from metallization steps on *metal1* or *metal2*.

  You use this model to calculate a partial antenna ratio (PAR). A PAR tells you if any single metallization step is likely to inflict damage to a gate.

- The cumulative checking model

  This model is more conservative than the partial checking model. It adds damage to a gate caused by the PAE referred to the gate from each metallization step, starting from *metal1* up to the layer that is being checked. For example, if you use the cumulative checking model to calculate the PAE referred to a gate from *metal3*, you add the PAR from the relevant antenna areas on *metal1*, *metal2*, and *metal3*.

  You use this model to calculate a cumulative antenna ratio (CAR). A CAR adds the damages on successive layers together to accumulate them as the layers are built up.

Calculating the Antenna Area

The area used to model the charge-collecting ability of a node is called the *antenna area*. The router calculates the antenna area for one of the following areas:

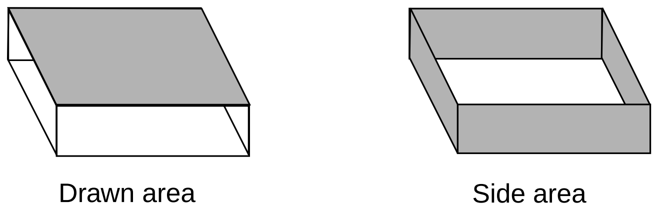
- The drawn area (the top surface area of the metal shape)

- The side area (the area of the sides of the metal shape)

The height of each side is taken from the `THICKNESS` statement for that layer.

Figure C-3 on page 260 shows drawn and side areas.

Figure C-3



Antenna Area Factor

You can increase or decrease the calculated antenna area by specifying an antenna area factor in the LEF file.

- Use `ANTENNAAREAFACTOR` to adjust the calculation of the drawn area.
- Use `ANTENNASIDEAREAFACTOR` to adjust the calculation of the side area.

The default value of both factors is `1`.

The final ratio check can be scaled (that is, made more or less pessimistic) by using the `ANTENNAAREAFACTOR` or `ANTENNASIDEAREAFACTOR` values that are used to multiply the final PAR and CAR values.

Note: The LEF and DEF `ANTENNA` values are always unscaled values; only the final ratio-check is affected by the scale factors.

Calculating a PAR

The formula to calculate a PAR is

$$PAR(N_{i,j},G_k) = \frac{Area(N_{i,j})}{\overset{G_k \in C(N_{i,j})}{\underset{G_k}{\sum}} Area(G_k)}$$

$PAR(N_{i,j}, G_k)$ is the partial antenna ratio for node j on $metal_i$ with respect to gate G_k, where G_k is electrically connected to node $N_{i,j}$ by layer i or below.

$Area(N_{i,j})$ is the drawn or side area of node $N_{i,j}$.

$C(N_{i,j})$ is the set of gates G_k that are electrically connected to $N_{i,j}$ through the layers below $metal_i$.

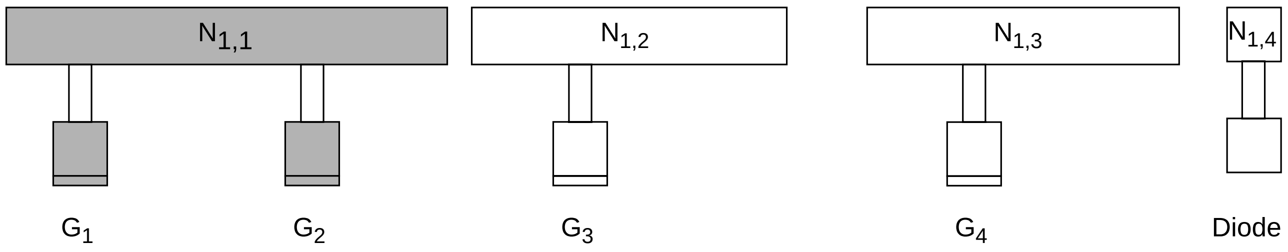
$Area(G_k)$ is the drawn or side area of gate G_k. (The reason to include the G_k parameter for PAR is to maintain uniformity with the notation for CAR.)

Note: For a specified node $N_{i,j}$, the $PAR(N_{i,j}, G_k)$ for all gates G_k that are connected to the node $N_{i,j}$ using $metal_i$ or below are identical.

Calculations for PAR on the First Metal Layer

Figure C-4 on page 261 shows a section of an imaginary chip after the first metal layer is processed.

Figure C-4



The shaded areas in the figure represent the wire segment and the gates whose areas you must compute to evaluate the formula below.

To calculate $PAR(N_{i,j}, G_k)$ for node $N_{1,1}$, a node on the first metal layer, with respect to gate G_1, use the following formula:

$$PAR(N_{1,1},G_1) = \frac{Area(N_{1,1})}{Area(G_1) + Area(G_2)}$$

Because gates G_1 and G_2 both connect to node $N_{1,1}$, the following statement is true:

$PAR(N_{1,1},G_1) = PAR(N_{1,1},G_2)$

To calculate PAR for node $N_{1,2}$, another node on the first metal layer, with respect to gate G_3, use the following formula:

$$\text{PAR}(N_{1,2},G_3) = \frac{\text{Area}(N_{1,2})}{\text{Area}(G_3)}$$

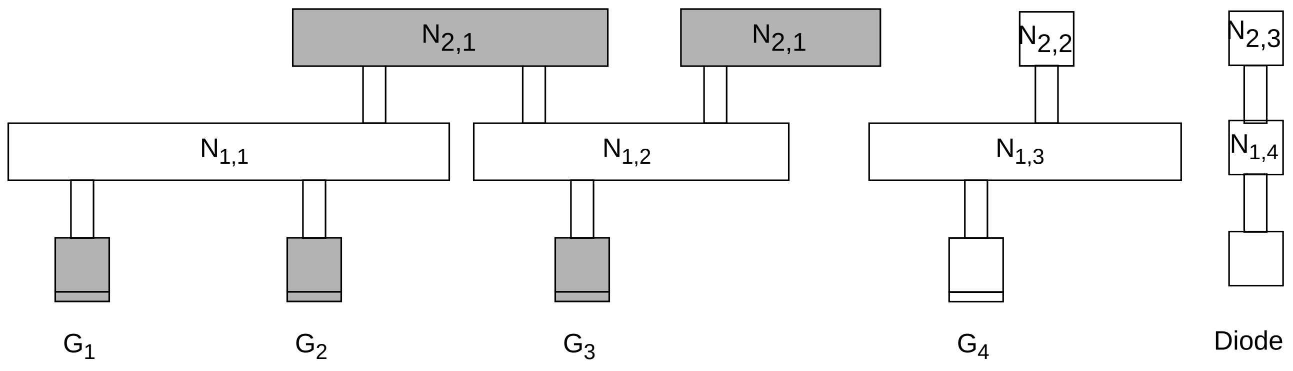
To calculate $\text{PAR}(N_{i,j}, G_k)$ for node $N_{1,3}$, another node on the first metal layer, with respect to gate G_4, use the following formula:

$$\text{PAR}(N_{1,3},G_4) = \frac{\text{Area}(N_{1,3})}{\text{Area}(G_4)}$$

Calculations for PAR on the Second Metal Layer

Figure C-5 on page 262 shows the chip after the second metal layer is processed.

Figure C-5



The shaded areas in the figure represent the wire segments and the gates whose areas you must compute to evaluate the formula below.

$N_{2,1}$ consists of two pieces of metal on the second layer that are electrically connected at this step in the fabrication process. Therefore, to calculate $\text{PAR}(N_{2,1},G_1)$, you must add the area of both pieces together.

To calculate `PAR(`$N_{i,j}$`,`G_k`)` for node $N_{2,1}$, a node on the second metal layer, with respect to gate G_1, use the following formula:

$$PAR\ (N_{2,1},G_1) = \frac{Area(N_{2,1})}{Area(G_1) + Area(G_2) + Area(G_3)}$$

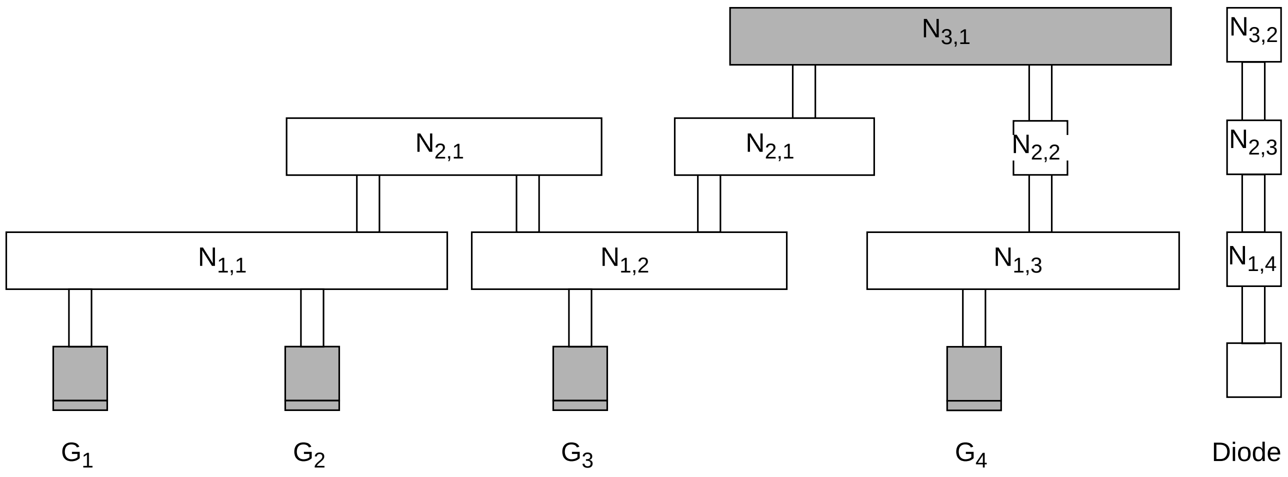
As on the first layer,

$PAR(N_{2,1},G_1) = PAR(N_{2,1},G_2) = PAR(N_{2,1},G_3)$

Calculations for PAR on the Third Metal Layer

Figure C-6 on page 263 shows the chip after the third metal layer is processed.

Figure C-6



The shaded areas in the figure represent the wire segment and the gates whose areas you must compute to evaluate the formula below.

To calculate `PAR(`$N_{i,j}$`, `G_k`)`for node $N_{3,1}$, a node on the third metal layer, with respect to gate G_1, use the following formula:

$$PAR\ (N_{3,1},G_1) = \frac{Area(N_{3,1})}{Area(G_1) + Area(G_2) + Area(G_3) + Area(G_4)}$$

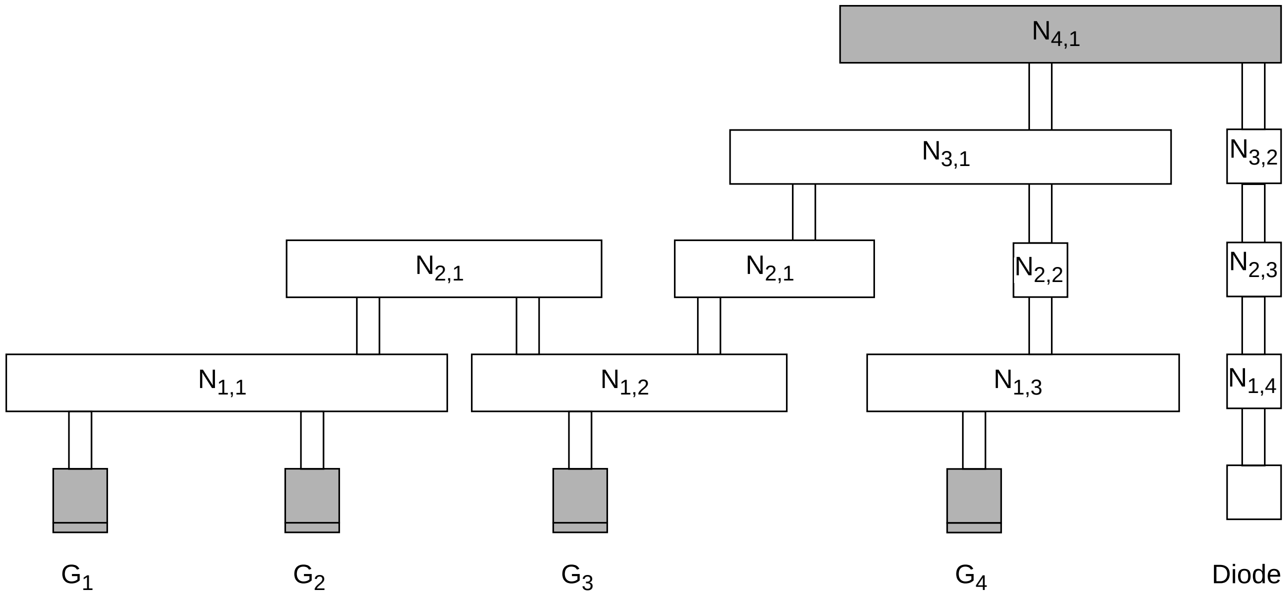
As on the prior layers,

$PAR(N_{3,1},G_1) = PAR(N_{3,1},G_2) = PAR(N_{3,1},G_3) = PAR(N_{3,1},G_4)$

Calculations for PAR on the Fourth Metal Layer

Figure C-7 on page 264 shows the chip after the fourth metal layer is processed.

Figure C-7



The shaded areas in the figure represent the wire segment and the gates whose areas you must compute to evaluate the formula below.

To calculate `PAR(`$N_{i,j}$`, `G_k`)` for the fourth metal layer, use the following formula:

$$\text{PAR }(N_{4,1},G_1) = \frac{\text{Area}(N_{4,1})}{\text{Area}(G_1) + \text{Area}(G_2) + \text{Area}(G_3) + \text{Area}(G_4)}$$

As on the prior layers,

$\text{PAR}(N_{4,1},G_1) = \text{PAR}(N_{4,1},G_2) = \text{PAR}(N_{4,1},G_3) = \text{PAR}(N_{4,1},G_4)$

Note: Node $N_{4,1}$ is connected to the diffusion layer through the output diode. After the router calculates the antenna ratio, it compares its calculations to the area of the diffusion, instead of the area of the gates.

Calculating a CAR

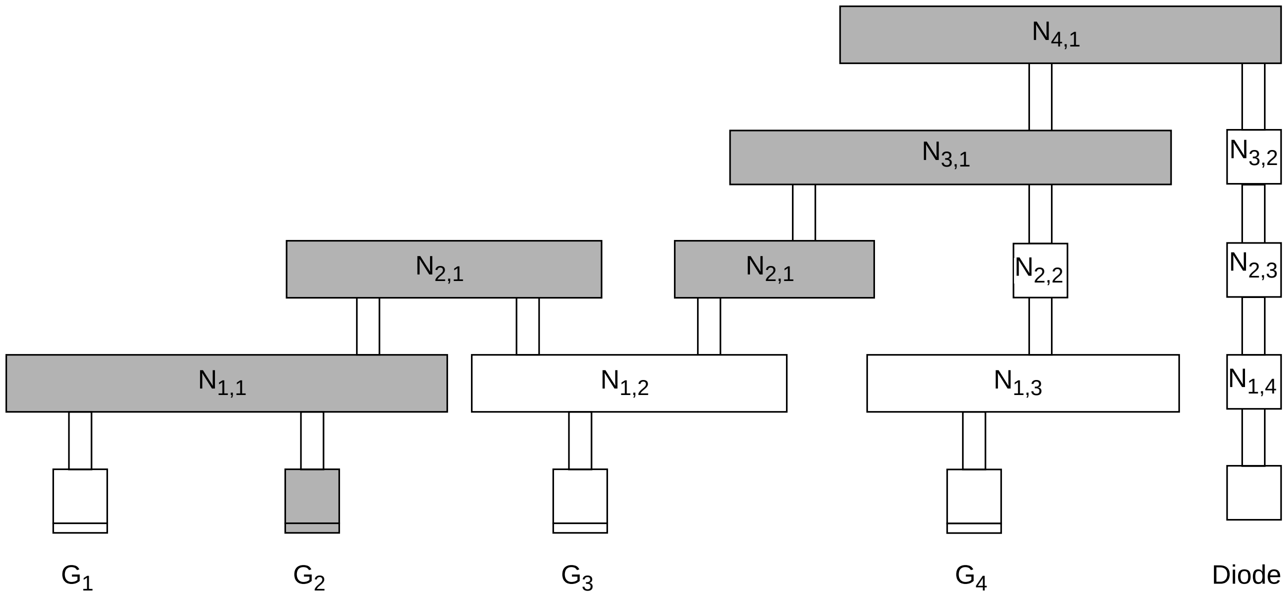
To calculate a CAR, the router adds the PARs for all the relevant nodes on the specified or lower metal layers that are electrically connected to a gate. Therefore, $\text{CAR}(N_{i,j},G_k)$

designates the cumulative damage to gate G_k by metallization steps up to the current level of metal, *i*.

Note: In practice, the router only needs to keep track of the worst-case CAR; however, the CARs for all of the gates shown in Figure C-8 on page 265 are described here.

The router calculates an antenna ratio with respect to a node-gate pair. To find the CAR for the node $N_{i,j}$ - gate G_k pair, you trace the path of the current between gate G_k and node $N_{i,j}$ and add the PAR with respect to gate G_k for the all nodes in the path between the first metal layer and layer *i* that you can trace back to G_k.

Figure C-8



The path of the current between gate G_2 and node $N_{4,1}$ is shaded.

Important

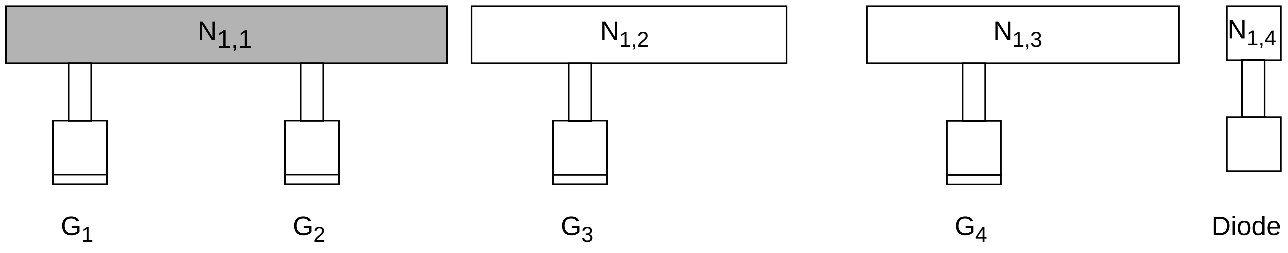
In Figure C-8 on page 265, node $N_{1,2}$ is not shaded because it was not electrically connected to G_2 when *metal1* was processed. That is, because the charge accumulated on $N_{1,2}$ when *metal1* was processed cannot damage gate G_1, the router does not include it in the calculations for $CAR(N_{2,1},G_1)$.

Another way to explain this is to say that the PAE from node $N_{1,2}$ with respect to gate G_2 is 0.

Calculations for CAR on the First Metal Layer

Figure C-9 on page 266 shows the chip after the first metal layer is processed.

Figure C-9



In the figure above,

$\mathrm{CAR}(N_{1,1},G_1) = \mathrm{PAR}(N_{1,1},G_1)$

$\mathrm{CAR}(N_{1,1},G_2) = \mathrm{PAR}(N_{1,1},G_2)$

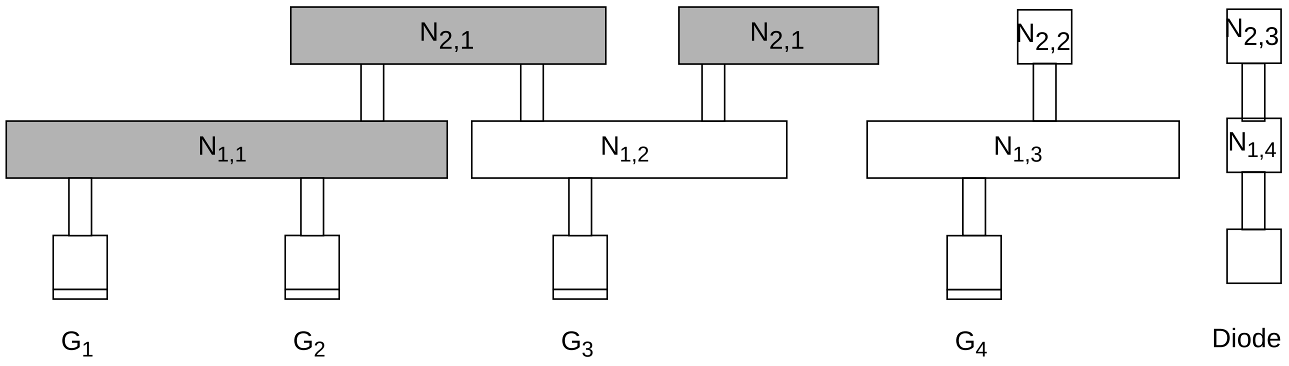
Because $\mathrm{PAR}(N_{1,1},G_1)$ equals $\mathrm{PAR}(N_{1,1},G_2)$, $\mathrm{CAR}(N_{1,1},G_1)$ equals $\mathrm{CAR}(N_{1,1},G_2)$.

Note: In general, $\mathrm{CAR}(N_{i,j},G_k)$ equals $\mathrm{CAR}(N_{i,j},G_{k'})$ if the two gates G_k and $G_{k'}$ are electrically connected to the same node on *metal1*, the lowest layer that is subject to the process antenna effect.

Calculations for CAR on the Second Metal Layer

Figure C-10 on page 267 shows the chip after the second metal layer is processed.

Figure C-10



The path of the current between gate G_1 and node $N_{2,1}$ is shaded.
Note that node $N_{2,1}$ comprises two pieces of metal.

Important

In the figure above, $N_{1,2}$ is not included in the calculations for $CAR(N_{2,1},G_1)$ because it was not electrically connected to G_1 when *metal1* was processed. That is, because the charge accumulated on $N_{1,2}$ when *metal1* was processed cannot damage gate G_1, the router does not include it in the calculations for $CAR(N_{2,1},G_1)$.

In the figure above,

$CAR(N_{2,1},G_1) = PAR(N_{1,1},G_1) + PAR(N_{2,1},G_1)$

$CAR(N_{2,1},G_2) = PAR(N_{1,1},G_2) + PAR(N_{2,1},G_2)$

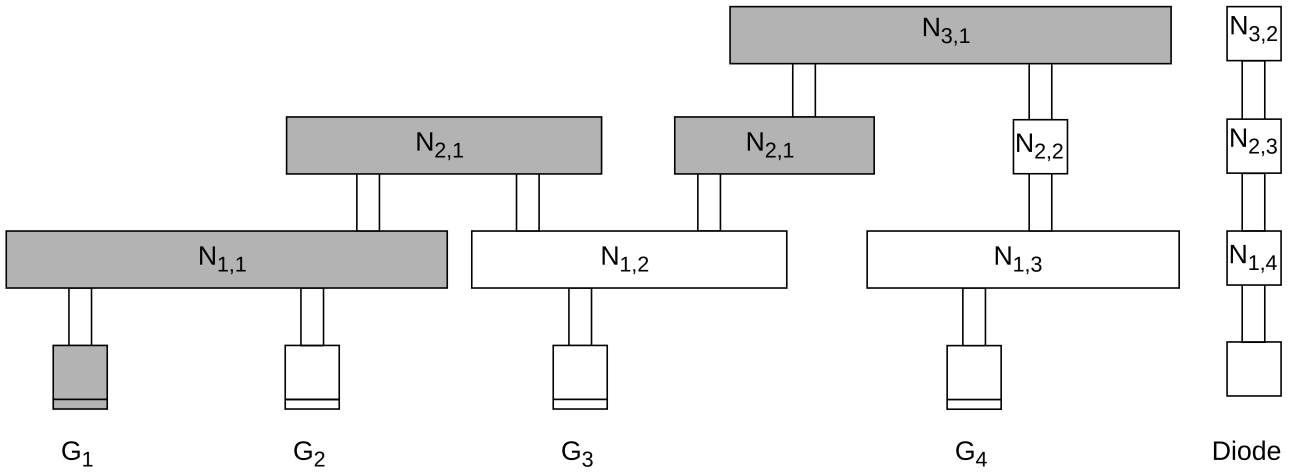
Gates G_1 and G_2 have the same history with regard to PAE because they are connected to the same piece of *metal1*, so they have the same CAR for any node on a specified layer:

$CAR(N_{2,1},G_1) = CAR(N_{2,1},G_2)$

Calculations for CAR on the Third Metal Layer

Figure C-11 on page 268 shows the chip after the third metal layer is processed.

Figure C-11



The path of the current between gate G_1 and node $N_{3,1}$ is shaded.

Gate G_1

In the figure above,

`CAR`$(N_{3,1},G_1)$ = `PAR`$(N_{1,1},G_1)$ + `PAR`$(N_{2,1},G_1)$ + `PAR`$(N_{3,1},G_1)$

Gate G_2

In the figure above,

`CAR`$(N_{3,1},G_2)$ = `PAR`$(N_{1,1},G_2)$ + `PAR`$(N_{2,1},G_2)$ + `PAR`$(N_{3,1},G_2)$

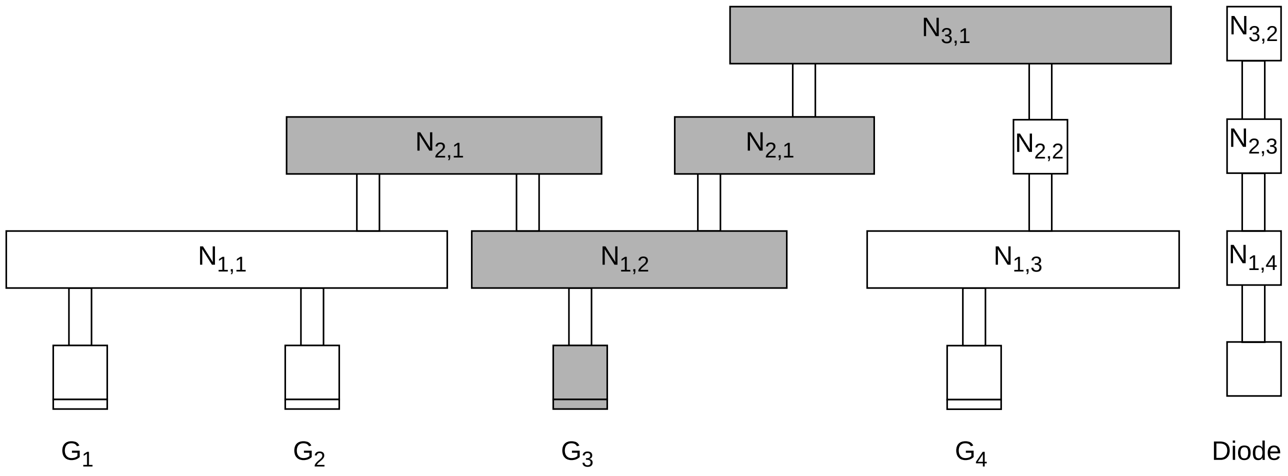
`CAR`$(N_{3,1},G_1)$ equals `CAR`$(N_{3,1},G_2)$ because gates G_1 and G_2 are both electrically connected to the same node, $N_{1,1}$, on *metal1* and therefore have the same history with regard to PAE. Therefore, the formula for `CAR`$(N_{3,2}, G_2)$ is `CAR`$(N_{3,1},G_1)$ = `CAR`$(N_{3,1},G_2)$

Gates G_3 and G_4

Gates G_3 and G_4 are not connected to the same node on *metal1* and therefore do not have the same history with regard to PAE. Therefore, the `CAR`$(N_{3,1},G_3)$ and `CAR`$(N_{3,1},G_4)$ do not necessarily equal `CAR`$(N_{3,1},G_1)$ or `CAR`$(N_{3,1},G_2)$.

In Figure C-12 on page 269, the relevant areas for calculating CAR for gate G_3 are shaded.

Figure C-12

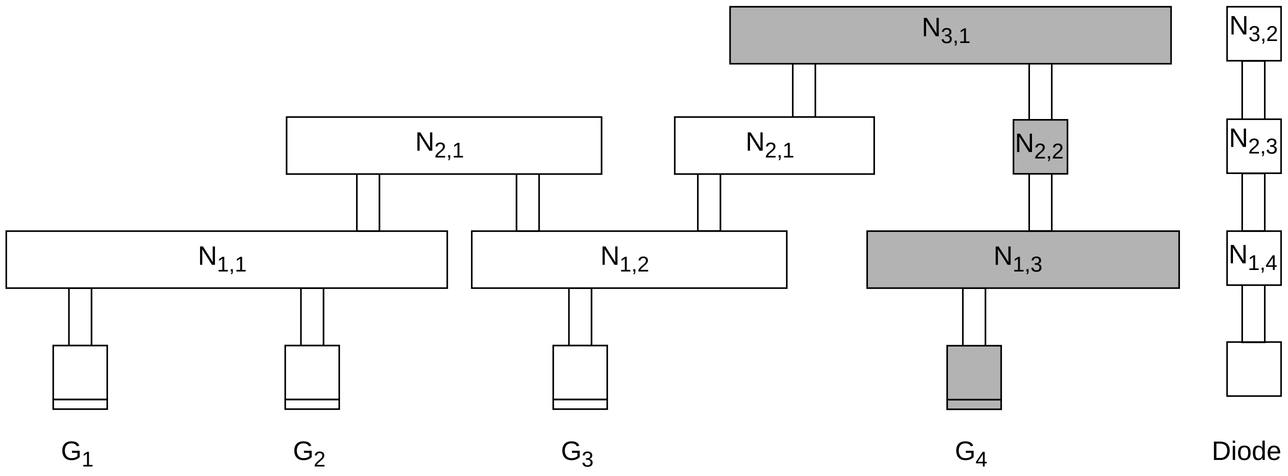


In the figure above,

`CAR(`$N_{3,1}$`,`G_3`) = PAR(`$N_{1,2}$`,`G_3`) + PAR(`$N_{2,1}$`,`G_3`) + PAR(`$N_{3,1}$`,`G_3`)`

In Figure C-13 on page 269, the relevant areas for calculating CAR for gate G_4 are shaded.

Figure C-13



In the figure above,

`CAR(`$N_{3,1}$`,`G_4`) = PAR(`$N_{1,3}$`,`G_4`) + PAR(`$N_{2,2}$`,`G_4`) + PAR(`$N_{3,1}$`,`G_4`)`

Calculations for CAR on the Fourth Metal Layer

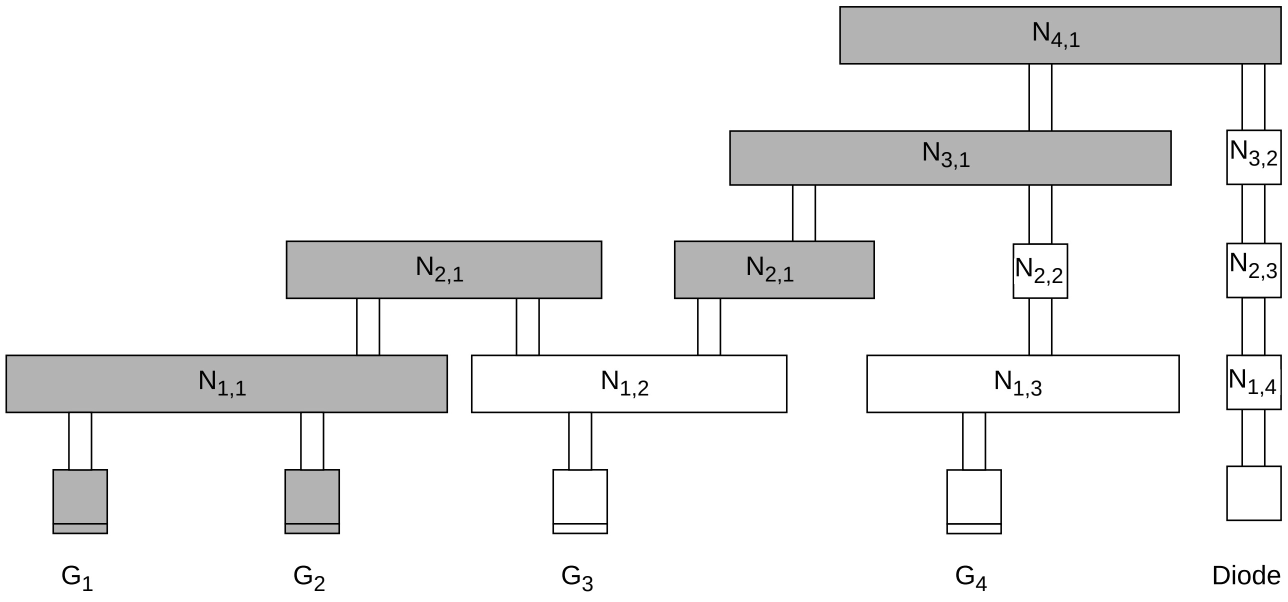
The following figure shows the chip after the fourth metal layer is processed.

Note: Node `N`$_{4,1}$ is connected to the diffusion layer through the output diode. After the router calculates the antenna ratio, it compares its calculations to the area of the diffusion, instead of the area of the gates.

Gates G_1 and G_2

In Figure C-14 on page 270, the relevant areas for calculating $CAR(N_{4,1},G_1)$ and $CAR(N_{4,1},G_2)$ are shaded.

Figure C-14



In the figure above,

$$CAR(N_{4,1},G_1) = PAR(N_{1,1},G_1) + PAR(N_{2,1},G_1) + PAR(N_{3,1},G_1) + PAR(N_{4,1},G_1)$$

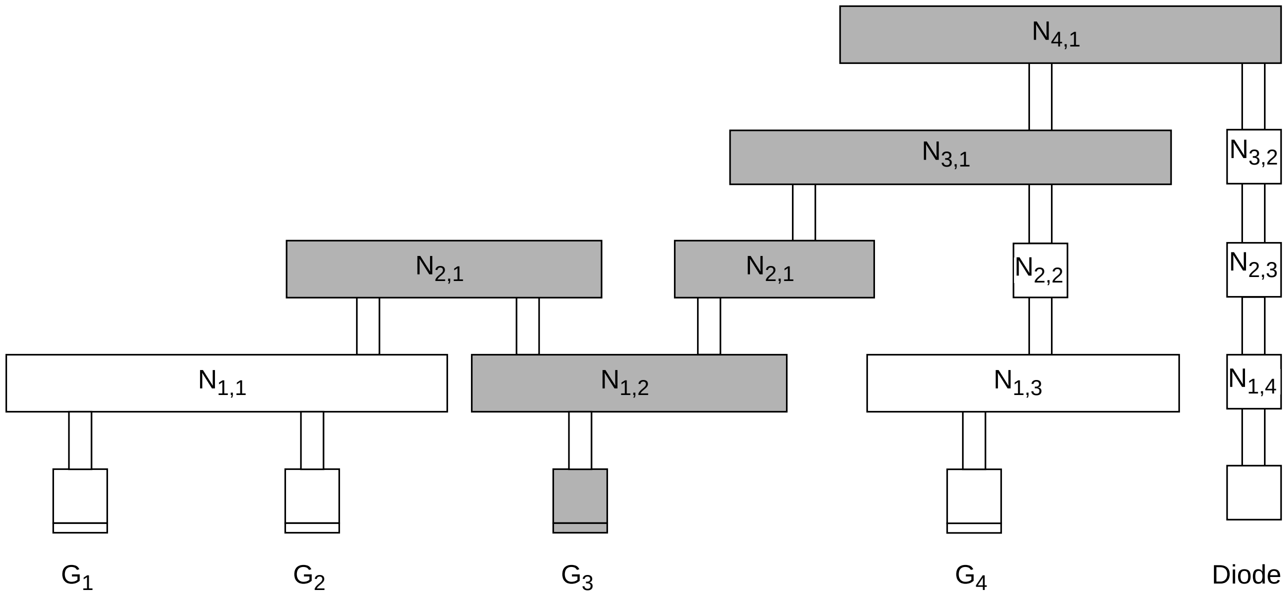
$$CAR(N_{4,1},G_2) = PAR(N_{1,1},G_2) + PAR(N_{2,1},G_2) + PAR(N_{3,1},G_2) + PAR(N_{4,1},G_2)$$

$$CAR(N_{4,1},G_1) = CAR(N_{4,1},G_2)$$

Gate G_3

In Figure C-15 on page 271, the relevant areas for calculating `CAR(`$N_{4,1}$`,`G_3`)` are shaded.

Figure C-15



In the figure above,

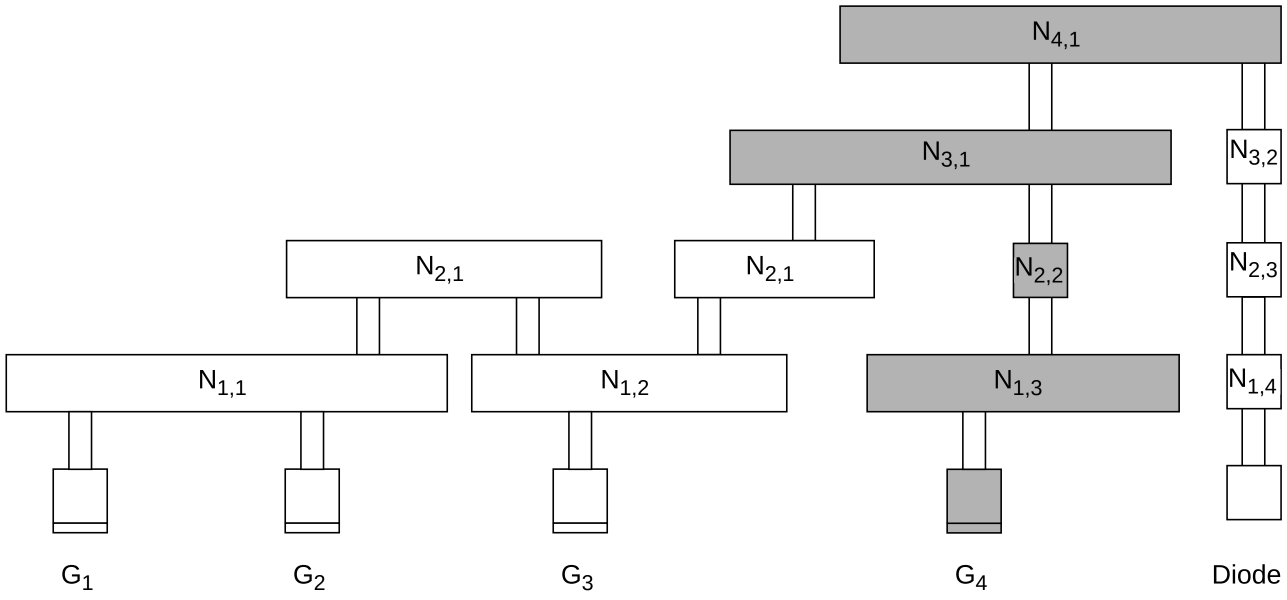
$$CAR(N_{4,1},G_3) = PAR(N_{1,2},G_3) + PAR(N_{2,1},G_3) + PAR(N_{3,1},G_3) + PAR(N_{4,1},G_3)$$

`CAR(`$N_{4,1}$`,`G_3`)` does not equal `CAR(`$N_{4,1}$`,`G_1`)` or `CAR(`$N_{4,1}$`,`G_2`)` because it is not connected to the same node on *`metal1`*.

Gate G_4

In Figure C-16 on page 272, the relevant areas for calculating `CAR(`$N_{4,1}$`,`G_4`)` are shaded.

Figure C-16



In the figure above,

```
CAR(N4,1,G4) = PAR(N1,3,G4) + PAR(N2,2,G4)
  + PAR(N3,1,G4) + PAR(N4,1,G4)
```

`CAR(N4,1,G4)` does not equal `CAR(N4,1,G1)`, `CAR(N4,1,G2)`, or `CAR(N4,1,G3)` because it is not connected to the same node on *metal1*.

Calculating Ratios for a Cut Layer

The router calculates damage from a cut layer separately from damage from a metal layer. Calculations for the cut layers do not use side area modelling.

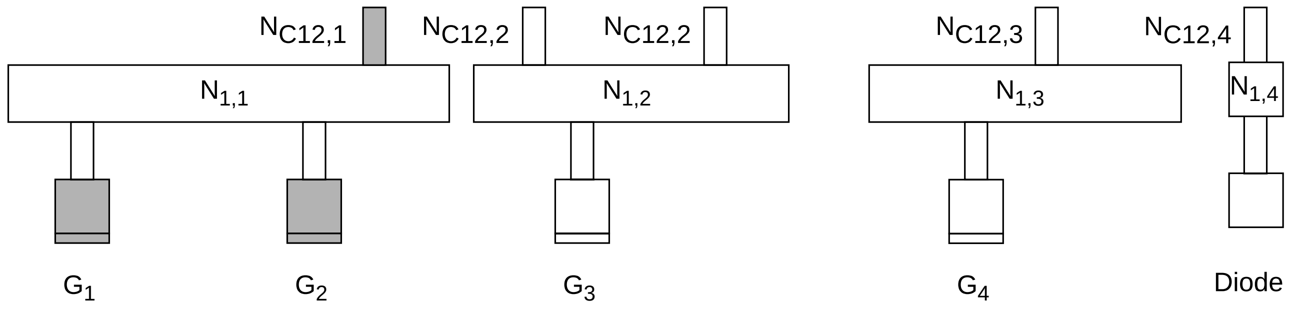
In the figures and text that follow,

- `Cij` is the cut layer between $metal_i$ and $metal_j$.
- $N_{Cij,k}$ specifies an electrically connected node on `Cij`.
- The nodes are numbered sequentially, from left to right.

Calculating a PAR on a Cut Layer

Figure C-17 on page 273 shows the chip after the `C12` process step.

Figure C-17



In the figure above,

$$PAR(N_{C12,1},G_1) = \frac{Area(N_{C12,1},G_1)}{Area(G_1) + Area(G_2)}$$

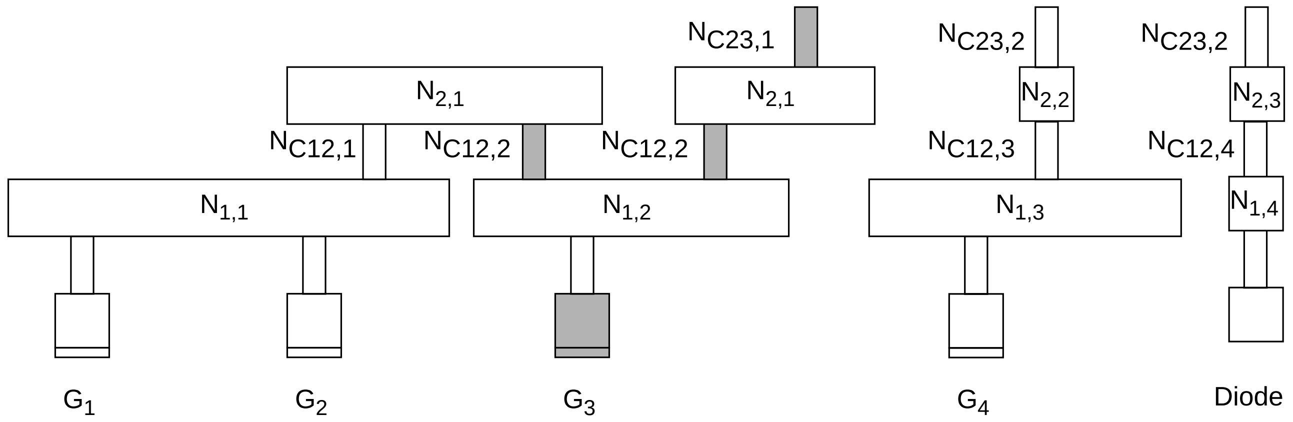
As in calculations on the metal layers,

$PAR(N_{C12,1},G_1) = PAR(N_{C12,2},G_2)$

Calculating a CAR on a Cut Layer

Figure C-18 on page 273 shows the chip after the `C23` process step.

Figure C-18

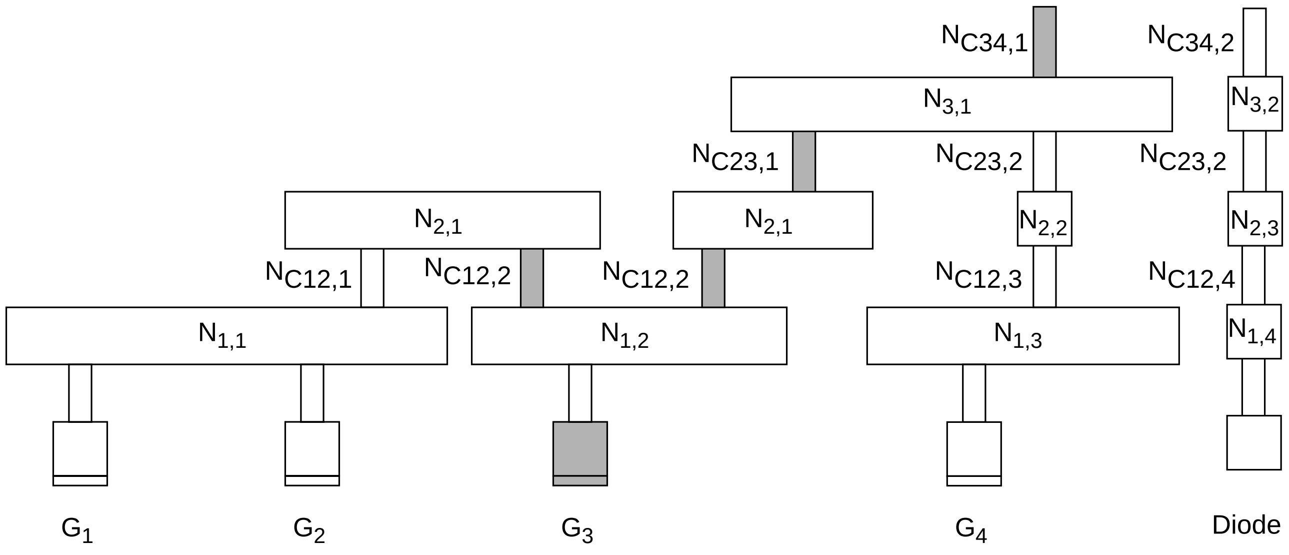


The router calculates the CAR with respect to gate `G`$_3$ after the cut `C23` process step as follows:

$$\mathrm{CAR}(N_{C23,1},G_3) = \frac{\mathrm{Area}(N_{C23,1},G_3)}{\mathrm{Area}(G_3)} + \frac{\mathrm{Area}(N_{C12,2},G_3)}{\mathrm{Area}(G_3)}$$

Figure C-19 on page 274 shows the chip after the `C34` process step.

Figure C-19



The router calculates the CAR with respect to gate `G`$_3$ after the cut `C34` process step as follows:

$$\mathrm{CAR}(N_{C34,1},G_3) = \frac{\mathrm{Area}(N_{C34,1},G_3)}{\mathrm{Area}(G_3)} + \frac{\mathrm{Area}(N_{C23,1},G_3)}{\mathrm{Area}(G_3)} + \frac{\mathrm{Area}(N_{C12,2},G_3)}{\mathrm{Area}(G_3)}$$

Checking for Antenna Violations

For each metal layer, the router performs several antenna checks, using the keywords and values specified in the LEF or DEF file. The router can perform the following four types of antenna checks, depending on the keywords you set in the LEF file:

- Area Ratio Check
- Side Area Ratio Check

- Cumulative Area Ratio Check
- Cumulative Side Area Ratio Check

Area Ratio Check

The area ratio check compares the PAR for each layer to the value of the `ANTENNAAREARATIO` or `ANTENNADIFFAREARATIO`.

The router calculates the PAR as follows:

$$\text{PAR}(N_{i,j}, G_k) = \frac{\text{Drawn area of } N_{i,j}}{\Sigma \text{ Area of gates connected below } N_{i,j}}$$

According to the formula above, the area ratio check finds the PAR for node $N_{i,j}$ with respect to gate G_k by dividing the drawn area of the node by the area of the gates that are electrically connected to it. The final PAR is multiplied by the `ANTENNAAREAFACTOR` (the default value for the factor is `1`) and compared to the `ANTENNAAREARATIO` or `ANTENNADIFFAREARATIO`. If the PAR is greater than the `ANTENNAAREARATIO` or `ANTENNADIFFAREARATIO` specified in the LEF file, the router finds a process antenna violation and attempts to fix it.

The link between $\text{PAR}(N_{i,j}, G_k)$ and a PAE violation at node $N_{i,j}$ depends on whether node $N_{i,j}$ is connected to a piece of diffusion, as follows:

- If there is no connection from node $N_{i,j}$ to a diffusion area through the current and lower layers, a violation occurs when the PAR is greater than the `ANTENNAAREARATIO`.
- If there is a connection from node $N_{i,j}$ to a diffusion area through current and lower layers, a violation occurs when the PAR is greater than the `ANTENNADIFFAREARATIO`.
- If there is a connection from node $N_{i,j}$ to a diffusion area through current and lower layers, and `ANTENNADIFFAREA` is not specified for an output or inout pin, the value is `0`.

Side Area Ratio Check

The side area ratio check compares the PAR computed based on the side area of the nodes for each layer to the value of the `ANTENNASIDEAREARATIO` or `ANTENNADIFFSIDEAREARATIO`.

The router calculates the PAR as follows:

$$\mathrm{PAR}(N_{i,j}\ G_k) = \frac{\text{Side area of } N_{i,j}}{\Sigma \text{ Area of gates connected below } N_{i,j}}$$

According to the formula above, the area ratio check finds the PAR for node $N_{i,j}$ with respect to gate G_k by dividing the side area of the node by the area of the gates that are electrically connected to $N_{i,j}$. The final PAR is multiplied by the `ANTENNASIDEAREAFACTOR` (the default value for the factor is `1`) and compared to the `ANTENNASIDEAREARATIO` or `ANTENNADIFFSIDEAREARATIO`. If the PAR is greater than the `ANTENNASIDEAREARATIO` or `ANTENNADIFFSIDEAREARATIO` specified in the LEF file, the router finds a process antenna violation and attempts to fix it.

The link between $\mathrm{PAR}(N_{i,j},G_k)$ and a PAE violation at node $N_{i,j}$ depends on whether node $N_{i,j}$ is connected to a piece of diffusion, as follows:

- If there is no connection to the diffusion area through the current and lower layers, a violation occurs when the PAR is greater than the `ANTENNASIDEAREARATIO`.
- If there is a connection to the diffusion area through current and lower layers, a violation occurs when the PAR is greater than the `ANTENNADIFFSIDEAREARATIO`.
- If there is a connection to the diffusion area through current and lower layers, and `ANTENNADIFFAREA` is not specified for an output or inout pin, the value is `0`.

Cumulative Area Ratio Check

The cumulative area ratio check compares the CAR to the value of `ANTENNACUMAREARATIO` or `ANTENNACUMDIFFAREARATIO`. The CAR is equal to the sum of the PARs of all nodes on the same or lower layers that are electrically connected to the gate.

Note: When you use CARs, you can ignore metal layers by not specifying the CAR keywords for those layers. For example, if you want to check *metal1* using a PAR and the remaining metal layers using a CAR, you can define `ANTENNAAREARATIO` or `ANTENNASIDEAREARATIO` for *metal1*, and `ANTENNACUMAREARATIO` or `ANTENNACUMSIDEAREARATIO` for the remaining metal layers.

The cumulative area ratio check finds the CAR for node $N_{i,j}$ with respect to gate G_k by adding the PARs for all layers of metal, from the current layer down to *metal1*, for all nodes that are electrically connected G_k. The final CAR is multiplied by the `ANTENNAAREAFACTOR` (the default value for the factor is `1`) and compared to the `ANTENNACUMAREARATIO` or `ANTENNACUMDIFFAREARATIO`. If the CAR is greater than the `ANTENNACUMAREARATIO` or

`ANTENNACUMDIFFAREARATIO` specified in the LEF file, the router finds a process antenna violation and attempts to fix it.

The link between $\mathtt{CAR}(\mathtt{N}_{i,j},\mathtt{G}_k)$ and a PAE violation at node $\mathtt{N}_{i,j}$ depends on whether node $\mathtt{N}_{i,j}$ is connected to a piece of diffusion, as follows:

- If there is no connection to a diffusion area through the current and lower layers, a violation occurs when the CAR is greater than the `ANTENNACUMAREARATIO`.
- If there is a connection to a diffusion area through current and lower layers, a violation occurs when the CAR is greater than the `ANTENNACUMDIFFAREARATIO`.
- If there is a connection to a diffusion area through current and lower layers, and `ANTENNADIFFAREA` is not specified for an output or inout pin, the value is `0`.

Cumulative Side Area Ratio Check

The cumulative side area ratio check compares the CAR to the value of the `ANTENNACUMSIDEAREARATIO` or `ANTENNACUMDIFFAREARATIO`.

Note: When you use CARs, you can ignore metal layers by not specifying the CAR keywords for those layers. For example, if you want to check *metal1* using a PAR and the remaining metal layers using a CAR, you can define `ANTENNAAREARATIO` or `ANTENNASIDEAREARATIO` for *metal1*, and `ANTENNACUMAREARATIO` or `ANTENNACUMSIDEAREARATIO` for the remaining metal layers.

The cumulative side area ratio check finds the CAR for node $\mathtt{N}_{i,j}$ with respect to gate $\mathtt{G}_k$ by adding the PARs for all layers of metal, from the current layer down to *metal1*, for all nodes that are electrically connected $\mathtt{G}_\mathtt{k}$. The final CAR is multiplied by the `ANTENNASIDEAREAFACTOR` (the default value for the factor is `1`) and compared to the `ANTENNACUMSIDEAREARATIO` or `ANTENNACUMDIFFAREARATIO`. If the CAR is greater than the `ANTENNACUMSIDEAREARATIO` or `ANTENNACUMDIFFAREARATIO` specified in the LEF file, the router finds a process antenna violation and attempts to fix it.

- If there is no connection to a diffusion area through the current and lower layers, a violation occurs when the CAR is greater than the `ANTENNACUMSIDEAREARATIO`.
- If there is a connection to a diffusion area through current and lower layers, a violation occurs when the CAR is greater than the `ANTENNACUMSIDEAREARATIO`.
- If there is a connection to a diffusion area through current and lower layers, and `ANTENNACUMDIFFAREA` is not specified for an output or inout pin, the value is `0`.

Example Using the Antenna Keywords

The following example is a portion of a LEF file that shows the antenna keywords for a process that has cumulative area ratio damage for metal and cut layers.

Assume you have the following antenna rules for your process:

1. A maximum cumulative metal to gate area ratio of 1000
2. If a diode of greater than .1 microns is connected to the metal, the maximum metal ratio is: ratio = diode_area x 2000 + 5000
3. A maximum cumulative via to gate area ratio of 20
4. If a diode of greater than .1 microns is connected to the via, the maximum via ratio is: ratio = diode_area x 200 + 100

The corresponding LEF file would include:

```
LAYER M1
  TYPE ROUTING ;
  ...
  ANTENNACUMAREARATIO 1000 ;
  ANTENNACUMDIFFAREARATIO
       PWL ( ( 0 1000 ) ( 0.099 1000 ) ( 0.1 5200 ) ( 100 205000 ) ) ;
END M1

LAYER VIA1
  TYPE CUT ;
  ...
  ANTENNACUMAREARATIO 20 ;
  ANTENNACUMDIFFAREARATIO
       PWL ( ( 0 20 ) ( 0.099 20 ) ( 0.1 120 ) ( 100 20100 ) ) ;
END VIA1
```

A typical standard cell that has only *M1* pins and routing inside of it would have:

```
MACRO INV1X
  CLASS CORE ;
  ...
  PIN IN
    DIRECTION INPUT ;
    ANTENNAGATEAREA .5 LAYER M1 ; # connects to 0.5 µm^2 poly gate
    ANTENNAPARTIALMETALAREA 1.0 LAYER M1 ; # has 1.0 µm^2 M1 area.
           # Note that it should not include the M1 pin area, just the M1 routing
```

```
          # area that is not included in the PIN shapes. In many cases, all of the
          # M1 routing is included in the PIN, so this value is 0, and not in the
          # LEF at all.
    ANTENNAMAXAREACAR 10.0 LAYER M1 ; # has 10.0 cumulative ratio so far.
          # This value can include area from internal poly routing if poly routing
          # damage is accumulated with the metal layers. It does not include
          # the area of the M1 pin area, just the M1 routing area that is not
          # included in the PIN shapes. If poly damage is not included, and all
          # of the M1 routing is included in the PIN, this value will be 0, and
          # not in the LEF at all.
    ...
  END IN
  PIN OUT
    DIRECTION OUTPUT ;
    ANTENNADIFFAREA .2 LAYER M1 ; # connects to 0.2 µm^2 difusion area
    ANTENNAPARTIALMETALAREA 1.0 LAYER M1 ; # has 1.0 µm^2 M1 area
    # No ANTENNAMAXAREACAR value because no internal poly gate is connected
    ...
  END OUT
END INV1X
```

Using Antenna Diode Cells

Routers generally use one of two methods to fix process antenna violations:

- Change the routing by breaking the metal layers into smaller pieces
- Insert antenna diode cells to discharge the current

Changing the Routing

One method routers use to fix antenna violations is to limit the charge that is collected through the metal nodes exposed to the plasma. To do this, it goes up one layer or pushes the routing down one layer whenever the process antenna ratio exceeds the ratio set in the LEF file.

The router changes the routing by disconnecting nets with antenna violations and making the connections to higher metal layers instead. It does not make the connections to lower layers. This method works because the top metal layer always completes the connection from the gate to the output drain area of the driver, which is a diode that provides a discharge path.

Inserting Antenna Diode Cells

The second method routers use to repair antenna violations is to insert antenna diode cells in the design. The electrical charges on the metal that connects to the diodes is then discharged through the diode diffusion layer and substrate. The router inserts the diode cells automatically.

The following example shows a LEF definition of an antenna diode cell, with the `CLASS CORE ANTENNACELL` and `ANTENNADIFFAREA` defined:

```
MACRO antenna1
    CLASS CORE ANTENNACELL ;
    ...
    PIN ANT1
        AntennaDiffArea 1.0 ;
        PORT
        LAYER metal1 ;
        RECT  0.190 2.380 0.470 2.660 ;
        END
    END ANT1
END antenna1
```

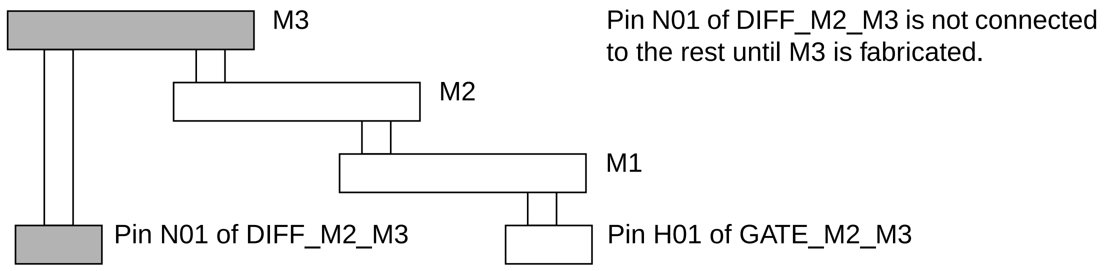
Using DiffUseOnly

LEF defines only one value for `ANTENNAAREAFACTOR` and one value for `ANTENNASIDEAREAFACTOR`, with or without `DIFFUSEONLY`, per layer. If you specify more than one antenna area or side area factor for a layer, only the last one is used. The `AREAFACTOR` value lets you scale the value of the metal area. If you use the `DIFFUSEONLY` keyword, only metal attached to diffusion is scaled.

Suppose you have the following LEF file:

```
Antenna.lef
-----------
LAYER M3
TYPE ROUTING ;
PITCH 0.56 ;
DIRECTION HORIZONTAL ;
WIDTH 0.28 ; SPACING 0.28 ;
SPACING 0.36 RANGE 1.0 250.0 ;
CAPACITANCE CPERSQDIST 0.0009762 ;
RESISTANCE RPERSQ 0.129 ;
THICKNESS 0.60 ;
AntennaAreaRatio 10000 ;
AntennaDiffAreaRatio 10000 ;
AntennaAreaFactor 1.2 DiffUseOnly ;
AntennaSideAreaRatio 5000 ;
AntennaDiffSideAreaRatio 5000 ;
AntennaSideAreaFactor 1.4 DiffUseOnly ;
END M3
```

Figure C-20


M3
Pin N01 of DIFF_M2_M3 is not connected to the rest until M3 is fabricated.
M2
M1
Pin N01 of DIFF_M2_M3
Pin H01 of GATE_M2_M3

In the figure,

- The input pin `H01` of `GATE_M2_M3` connects the metal wires to *metal1*, *metal2*, and *metal3* in sequence.
- The `ANTENNAAREAFACTOR 1.2 DIFFUSEONLY` and `ANTENNASIDEAREAFACTOR 1.4 DIFFUSEONLY` apply to *metal3* routing.
- Prior to *metal3* fabrication, there is no path to the diffusion diode. This causes the default factor of `1.0` to apply to the *metal1* and *metal2* segments shown when calculating PARs.

Calculations for Hierarchical Designs

The following section illustrates computation of antenna ratios for hierarchical designs.

LEF and DEF Keywords for Hierarchical Designs

If the keyword ends with ...	**It refers to ...**	**Examples**

`area`
`sideArea`
	Drawn area or side area of the metal wires. Measured in square microns.	`ANTENNAPARTIALCUTAREA`
		`ANTENNAPARTIALMETALAREA`
		`ANTENNAPARTIALMETALSIDEAREA`
		`ANTENNAPINDIFFAREA`
		`ANTENNAPINGATEAREA`
		`ANTENNAPINPARTIALCUTAREA`

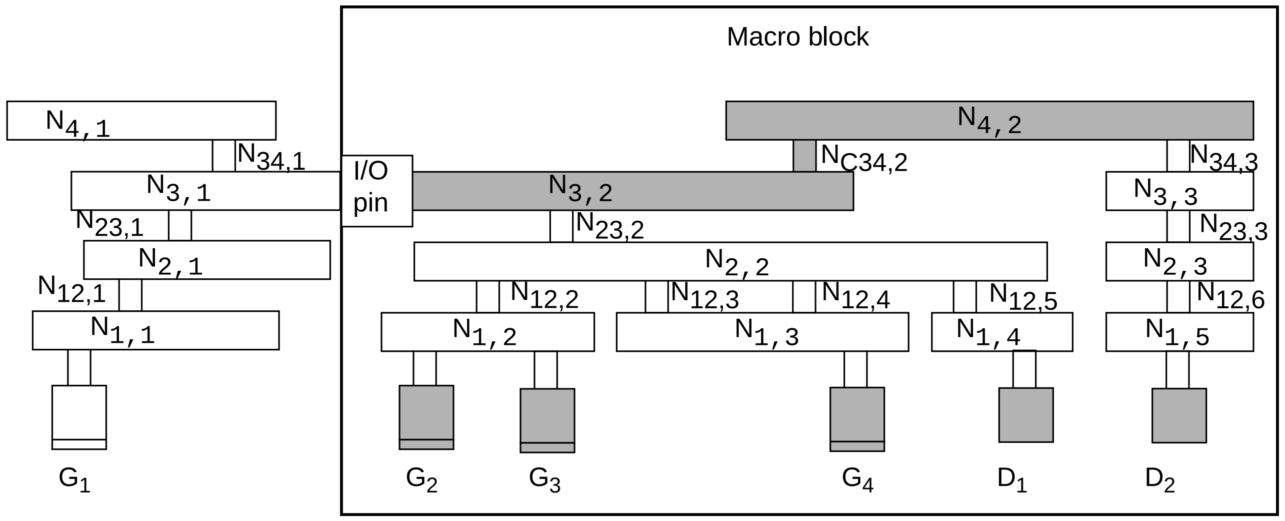
`CAR`	Relationship the router is calculating
	`CAR` is used in keywords for cumulative antenna ratio.
		`ANTENNAMAXAREACAR`
		`ANTENNAMAXCUTCAR`
		`ANTENNAMAXSIDEAREACAR`
		`ANTENNAPINMAXAREACAR`
		`ANTENNAPINMAXCUTCAR`
		`ANTENNAPINMAXSIDEAREACAR`

Design Example

Figure C-21 on page 283 represents a macro block. This block can be a custom hard block or part of a bottom-up hierarchical flow. The resulting PAE values will be the same in either case. In the example,

- Gates G_1, G_2, G_3, and G_4 are the same size.
- Node $N_{1,3}$ is larger than node $N_{1,2}$.
- Vias (cuts) are all the same size.
- The I/O pin is on *metal3*.
- The area of diffusion for D_1 is $area(Diff_1)$.
- The area of diffusion for D_2 is $area(Diff_2)$.
- The area of the cut layer that connects node $N_{3,1}$ and node $N_{4,2}$ is $area(N_{C34,1})$.
- Any damage from the poly layer or poly-to-metal1 via is ignored.

Figure C-21



Relevant Metal Areas

- The relevant metal area for PAE calculations is the partial metal drawn area and side area connected directly to the I/O pin on the inside of the macro on the specified layer.
- Only the same metal layer as the I/O pin or above is needed for PAR calculations in hierarchical designs.

Important

Do not include the drawn area or side area of the I/O pin in the area calculations for the block, because the router includes these areas in the calculations for the upper level. Only the internal routing area that is not part of the I/O pin should be included.

For the design in the figure above, you must specify values for the following metal areas in the LEF file:

ANTENNAPARTIALMETALAREA *area*($N_{3,2}$) LAYER Metal3 ;
ANTENNAPARTIALMETALAREA *area*($N_{4,2}$) LAYER Metal4 ;
ANTENNAPARTIALMETALSIDEAREA *sideArea*($N_{3,2}$) LAYER Metal3 ;
ANTENNAPARTIALMETALSIDEAREA *sideArea*($N_{4,2}$) LAYER Metal4 ;

You do not need to specify an `ANTENNAPARTIALMETALAREA` or `ANTENNAPARTIALSIDEMETALAREA` for any layer lower than *metal3* because the I/O pin is on *metal3*; that is, there is no connection outside the block until *metal3* is processed.

Relevant Gate, Diffusion, and Cut Areas

- The relevant gate and diffusion areas are the gate and diffusion areas that connect directly to the I/O pin on the specified layer or are electrically connected to the pin through lower layers.
- The relevant partial cut area is above the current pin layer and inside the macro on the specified layer.

For the design in the figure above, you must specify values for the following gate, diffusion, and cut areas in the LEF file:

`ANTENNAGATEAREA` *`area`*`(`G_2 + G_3 + G_4`) LAYER Metal3 ;`
`ANTENNADIFFAREA` *`area`*`(`Diff_1`) LAYER Metal3 ;`
`ANTENNADIFFAREA` *`area`*`(`Diff_1 + Diff_2`) LAYER Metal4 ;`
`ANTENNAPARTIALCUTAREA` *`area`*`(`$N_{34,2}$`) LAYER Via34 ;`

Calculating the CAR

Use the following keywords to calculate the actual CAR on the I/O pin layer or above.

- The relevant maximum CAR value of the drawn and side areas are from the metal layer that is on or below the I/O pin layer.
- The relevant maximum CAR value of the cut layer is from the cut layer that is immediately above the I/O pin layer.

For the example in Figure C-21 on page 283, the keywords and calculations for *metal3* and *via34* would be:

$$\text{ANTENNAMAXAREACAR}\left(\frac{N_{1,3}}{G_4}+\frac{N_{2,2}}{G_2+G_3+G_4}+\frac{N_{3,2}}{G_2+G_3+G_4}\right)\ \text{LAYER Metal3 ;}$$

$$\text{ANTENNAMAXSIDEAREACAR}\left(\frac{N_{1,3}}{G_4}+\frac{N_{2,2}}{G_2+G_3+G_4}+\frac{N_{3,1}}{G_2+G_3+G_4}\right)\ \text{LAYER Metal3 ;}$$

$$\text{ANTENNAMAXCUTCAR}\left(\frac{N_{12,3}+N_{12,4}}{G_4}+\frac{N_{23,2}}{G_2+G_3+G_4}+\frac{N_{34,2}}{G_2+G_3+G_4}\right)\ \text{LAYER Via34;}$$

Sample LEF File for a Bottom-Up Hierarchical Design

For a macro block like that shown in Figure C-21 on page 283, you should have the following pin information in your LEF file, ignoring `SIDEAREA` values:

```
PIN example
    ANTENNAGATEAREA 0.3 LAYER METAL3 ; # area of $G_2 + G_3 + G_4$
    ANTENNADIFFAREA 1.0 LAYER METAL3 ; # area of $D_1$
    ANTENNAPARTIALMETALAREA 10.0 LAYER METAL3 ; # area of $N_{3,2}$
    ANTENNAMAXAREACAR 100.0 LAYER METAL3 ; # max CAR of $N_{3,2}$

    ANTENNAPARTIALCUTAREA 0.1 LAYER VIA34 ; # area of $N_{34,2}$
    ANTENNAMAXCUTCAR 5.0 LAYER VIA34 ; # max cut CAR of $N_{34,2}$

    ANTENNAGATEAREA 0.3 LAYER METAL4 ; # area of $G_2 + G_3 + G_4$
    ANTENNADIFFAREA 2.0 LAYER METAL4 ; # area of $D_1 + D_2$
    ANTENNAPARTIALMETALAREA 12.0 LAYER METAL4 ; # area of $N_{4,2}$
    ANTENNAMAXAREACAR 130.0 LAYER METAL4 ; # max CAR of $N_{4,2}$
END example
```

Top-Down Hierarchical Design Example

In a top-down design, the router uses the top-level antenna values to check for process antennas inside the block. If the top level is routed first, the top-level routing CAR and PAR values can be passed down into the DEF for the sub-block. This method can also be used to pass down estimated “budgets” for PAR and CAR values.

Set the following keywords in the DEF file for the design. In a top-down design you assign a value to the I/O pin that indicates how much routing, CAR, and PAR occurred outside the block already.

```
MACRO macroName
    CLASS BLOCK ;
        PIN pinName
        DIRECTION OUTPUT ;
        [ANTENNAPINPARTIALMETALAREA value [LAYER layerName] ;] ...
        [ANTENNAPINPARTIALMETALSIDEAREA value [LAYER layerName] ;] ...
        [ANTENNAPINGATEAREA value [LAYER layerName] ;] ...
        [ANTENNAPINDIFFAREA value [LAYER layerName] ;] ...
        [ANTENNAPINMAXAREACAR value [LAYER layerName] ;] ...
        [ANTENNAPINMAXSIDEAREACAR value [LAYER layerName] ;] ...
        [ANTENNAPINPARTIALCUTAREA value [LAYER cutlayerName] ;] ...
        [ANTENNAPINMAXCUTCAR value LAYER cutlayerName] ;] ...
    END Z
END macroName
```

Sample DEF File for a Top-Down Hierarchical Design

An example of the DEF keywords for Figure C-21 on page 283 would be:

```
PINS 100 ;
    - example + NET example1
        + ANTENNAPINPARTIALMETALAREA ($N_{3,1}$) LAYER Metal3 ;
        + ANTENNAPINPARTIALMETALSIDEAREA ($N_{3,1}$) LAYER Metal3 ;
        + ANTENNAPINGATEAREA ($G_1$) LAYER Metal3 ;
         # No ANTENNAPINDIFFAREA for this example

       + ANTENNAPINMAXAREACAR $\left(\frac{N_{1,1}+N_{2,1}+N_{3,1}}{G_1}\right)$ LAYER Metal3 ;
       + ANTENNAPINMAXSIDEAREACAR $\left(\frac{N_{1,1}+N_{2,1}+N_{3,1}}{G_1}\right)$ LAYER Metal3 ;
        + ANTENNAPINPARTIALCUTAREA ($N_{34,1}$) LAYER via34 ;

       + ANTENNAPINMAXCUTCAR $\left(\frac{N_{34,1}+N_{23,1}+N_{12,1}}{G_1}\right)$ LAYER Metal3 ;
        + ANTENNAPINGATEAREA ($G_1$) LAYER Metal4 ;
        + ANTENNAPINPARTIALMETALAREA ($N_{4,1}$) LAYER Metal4 ;
        + ANTENNAPINPARTIALMETALSIDEAREA ($N_{4,1}$) LAYER Metal4 ;
        ...
END PINS
```

Index