MESH

The **MESH** statement initiates the mesh generation or reads a previously generated mesh.

```
MESH

 Initial Mesh Generation
   {  ( [ {RECTANGU | CYLINDRI} ] [DIAG.FLI] )

 Mesh File Input
    | ( IN.FILE=<c> [PROFILE]
        [ {  ASCII.IN
           | ( TSUPREM4 [ ELEC.BOT [Y.TOLER=<n>] ] [POLY.ELE]
               [X.MIN=<n>] [X.MAX=<n>] [Y.MIN=<n>] [Y.MAX=<n>]
               [FLIP.Y] [SCALE.Y=<n>]
             )
           | ( TIF [ ELEC.BOT [Y.TOLER=<n>] [POLY.ELE] ] )
          }
        ]
 New Automatic Boundary Conforming Mesh Generation
       [{ ( ABC
              [GRIDTOP] [VOIDELEC] [RFN.CRNR] [JUNC.ABC]
              [CRITICAL=<n>] [SPACING=<n>]
              [N.SEMICO=<n>] [N.INSULA=<n>] [N.CONDUC=<n>]
              [NORMGROW=<n>] [LATERAL=<n>] [ELIMINAT]
            ) |
 Old Automatic Boundary Conforming Mesh Generation
          ( ABC.OLD
              [SPACING=<n>][RATIO=<n>] [ANGLE=<n>]
              [MAX.SPAC=<n>] [CORNER=<n>] [LAYERS=<n>]
              [ATTEMPTS=<n>] [RELAX=<n>] [ELIMINAT]
              [EXTERNAL=<n>] [OPTIMIZE]   [JUNC.ABC]
              [N.SEMICO=<n>] [N.INSULA=<n>] [N.CONDUC=<n>]
              [N.SILICO=<n>] [N.POLYSI=<n>] [N.OXIDE=<n>]
              [N.NITRID=<n>] [N.OXYNIT=<n>] [N.SAPPHI=<n>]
              [N.BPSG=<n>]   [N.INAS=<n>]   [N.GAAS=<n>]
              [N.ALGAAS=<n>] [N.HGCDTE=<n>] [N.S.OXID=<n>]
              [N.SIC=<n>]    [N.INGAAS=<n>] [N.INP=<n>]
              [N.GERMAN=<n>] [N.DIAMON=<n>] [N.ZNSE=<n>]
              [N.ZNTE=<n>]   [N.A-SILI=<n>] [N.SIGE=<n>]
          )
        }
       ]
      )
   }
```

(**MESH** statement continued on next page)

(**MESH** statement continued from the previous page)

```
Mesh Adjustments
  [PERIODIC [PBC.TOL=<n>] ] [ORDER] [ADJUST] [VIRTUAL]
  [CENTROID] [ WIDTH=<n> N.SPACES=<n> [X.SPLIT=<n>] ]
  [OBTUSE.A=<n>]

Mesh File Output
  [ OUT.FILE=<c> [ASCII.OU] [NO.TTINF] ] [SMOOTH.K=<n>]
```

Parameter	Type	Definition	Default	Units
Initial Mesh Generation				
RECTANGU	logical	Specifies that the simulation mesh uses rectangular coordinates.	true	
CYLINDRI	logical	Specifies that the simulation mesh uses cylindrical coordinates. If this parameter is specified, the horizontal axis represents the radial direction and the vertical axis represents the z-direction.	false	
DIAG.FLI	logical	Specifies that the direction of diagonals is changed about the horizontal center of the grid. If this parameter is false, all diagonals are in the same direction.	true	
Mesh File Input				
IN.FILE	char	The identifier for the file containing a previously generated mesh. Unless **ASCII.IN** or **TSUPREM4** is specified, this file is in binary format. **synonym: INFILE**	none	
PROFILE	logical	Specifies that impurity profiles are input from the data file.	true	
ASCII.IN	logical	Specifies that the input mesh is stored in a formatted file.	false	
TSUPREM4	logical	Specifies that the input mesh was generated by TSUPREM-4 or by an external grid editor.	false	
ELEC.BOT	logical	Specifies that an electrode is added to the structure at the maximum y coordinate value.	false	
Y.TOLER	number	The maximum distance by which the y coordinate of a node can deviate from the maximum y coordinate value in the device and still be considered part of an electrode that is added using the **ELEC.BOT** parameter. This is useful if the bottom edge of the device is non-planar. This parameter is valid only if **TSUPREM4** is specified.	0	microns
POLY.ELE	logical	Specifies that regions defined as polysilicon in the data file are treated as electrodes.	true	
X.MIN	number	The minimum x coordinate read in from the data file. All nodes and elements with x coordinates less than the value specified by **X.MIN** are eliminated from the structure. This parameter is valid only if **TSUPREM4** is specified.	The minimum x location in the structure.	microns
X.MAX	number	The maximum x coordinate read in from the data file. All nodes and elements with x coordinates greater than the value specified by **X.MAX** are eliminated from the structure. This parameter is valid only if **TSUPREM4** is specified.	The maximum y location in the structure.	microns

Parameter	Type	Definition	Default	Units
Y.MIN	number	The minimum y coordinate read in from the data file. All nodes and elements with y coordinates less than the value specified by **Y.MIN** are eliminated from the structure. This parameter is valid only if **TSUPREM4** is specified.	The minimum y location in the structure.	microns
Y.MAX	number	The maximum y coordinate read in from the data file. All nodes and elements with y coordinates greater than the value specified by **Y.MAX** are eliminated from the structure. This parameter is valid only if **TSUPREM4** is specified.	The maximum y location in the structure.	microns
FLIP.Y	logical	Specifies that the direction of the vertical coordinate is reversed when the file is read. This parameter is valid only if **TSUPREM4** is specified.	false	
SCALE.Y	number	The scale factor to multiply all coordinate values by when reading the mesh file. This parameter is valid only if **TSUPREM4** is specified.	none	none
TIF	logical	Specifies that the file to be read in is in the TIF (Technology Interchange Format).	false	

New Automatic Boundary Conforming Mesh Generator

Parameter	Type	Definition	Default	Units
ABC	logical	Specifies that the device structure imported from the input file is remeshed using the new automatic boundary conforming mesh generator.	false	
GRIDTOP	logical	Grid the top boundary of the device.	true	
VOIDELEC	logical	Mesh volume electrodes without introducing any interior nodes.	false	
RFN.CRNR	logical	Automatically refine the grid spacing on region boundaries near corners where two or more regions meet.	true	
JUNC.ABC	logical	Specifies that the mesh should conform to junctions in addition to boundaries.	false	none
CRITICAL	number	Maximum allowed deviation of the region boundaries in the new mesh from those in the original mesh.	.001*minimum of device width and device height	microns
SPACING	number	Desired grid spacing along the boundaries and interfaces. The actual mesh spacing along the boundaries and interfaces may be smaller in order to satisfy the **CRITICAL** parameter.	(device width) / 50	microns
N.SEMICO	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for all semiconductor regions.	0.1	none
N.INSULA	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for all insulator regions.	1.0	none
N.CONDUC	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for all conductor regions.	.2	none
NORMGROW	number	Multiplication factor for the grid spacing normal to the region boundary. The element size grows in the normal direction from the region boundary toward the inside of the region. Each next grid spacing in the direction toward the inside of the region is larger than the previous spacing by the specified factor. If the factor is one, the mesh spacing is uniform throughout the region. The factor should be larger than one to obtain a coarser mesh away from the region boundaries. **synonym: RATIO**	1.2	none

Parameter	Type	Definition	Default	Units
LATERAL	number	Target lateral spacing during region meshing. This parameter is used to indicate the desired lateral spacing during the interior meshing of the regions. The actual lateral spacing obtained during meshing will be refined or unrefined as necessary in order to stay close to the specified value. Refinement or unrefinement of the lateral spacing will only occur, however, if non-obtuse elements can be produced. Very large values of **LATERAL** lead to unrefinement whenever non-obtuse elements can be produced. This is equivalent to using the **ELIMINAT** parameter.	maximum of device width and device height	microns
ELIMINAT	logical	Minimizes the number of nodes in the mesh by eliminating interior nodes without creating obtuse elements. **synonym: COARSEN**	true	

Old Automatic Boundary Conforming Mesh Generation

Parameter	Type	Definition	Default	Units
ABC.OLD	logical	Specifies that the device structure imported from the input file is remeshed using the automatic boundary conforming mesh generator.	false	
SPACING	number	Desired grid spacing along the boundaries and interfaces. The actual mesh spacing along the boundaries and interfaces can be smaller due to surface curvature. **synonym: STEP**	(device width) / 50	microns
RATIO	number	Multiplication factor for the grid spacing normal to the region boundary. The element size grows in the normal direction from the region boundary toward the inside of the region. Each next grid spacing in the direction toward the inside of the region is larger than the previous spacing by the specified factor. If the factor is one, the mesh spacing is uniform throughout the region. The factor should be larger than one to obtain a coarser mesh away from the region boundaries. **synonym: FACTOR**	1.2	none
ANGLE	number	Threshold angle for geometry smoothing. Allows unwanted small zig-zag type geometry features in the imported structure to be removed if deviation from a straight line is less than the specified threshold angle for any pair of adjacent edges along the region's boundary.	30	degrees
RELAX	number	Relaxation factor for mesh refinement near curved boundaries. The mesh along curved region boundaries and interfaces is automatically refined to avoid obtuse elements. A **RELAX** factor value of less than one guarantees no obtuse elements near the boundaries, but the mesh is harder to generate. On the contrary, a **RELAX** factor value larger than one allows some obtuse elements near the boundaries, but the mesh is easier to generate. **synonym: LOOSE**	1	none
MAX.SPAC	number	Maximum allowed grid spacing in the normal direction to the region boundaries. **synonym: MAX.STEP**	50	microns
ELIMINAT	logical	Minimizes the number of nodes in the mesh by eliminating interior nodes without creating obtuse elements. **synonym: COARSEN**	true	

Parameter	Type	Definition	Default	Units
CORNER	number	Corner smoothing factor for the interior boundary conforming grid lines. If this factor is zero, the interior grid lines adjacent to the region's boundary maintain the boundary shape. If it is nonzero, then each subsequent grid line adjacent to the boundary is smoother than the previous one (the curvature of the "corner" is reduced). The corner smoothing factor should be larger or equal to zero and smaller than one.	0.9	none
LAYERS	number	Number of allowed mesh "layers" of the boundary conforming type before switching to the unstructured Delaunay mesh in the region's interior.	500	none
ATTEMPTS	number	Number of allowed attempts to mesh the device when the meshing fails for the specified set of meshing parameters. Each subsequent attempt "relaxes" the meshing parameters.	4	none
EXTERNAL	number	Ratio of lateral spacing along external boundaries of the structure to the lateral spacing along internal boundaries (which is specified by the parameter **SPACING**). It is recommended that the value of the ratio is larger than one and less than ten in order to create a good mesh with minimum nodes on external boundaries.	4.0	none
OPTIMIZE	logical	Optimizes mesh quality by flipping all mesh edges whenever it improves quality of the adjacent triangles.	true	none
JUNC.ABC	logical	Specifies that the mesh should conform to junctions in addition to boundaries.	false	none
N.SEMICO	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for all semiconductor regions.	0.1	none
N.INSULA	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for all insulator regions.	0.1	none
N.CONDUC	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for all conductor regions. **synonyms: N.ELECTR, N.ALUMIN, N.TERMIN, N.METAL**	0.2	none
N.SILICO	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for silicon.	0.1	none
N.POLYSI	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for polysilicon.	0.2	none
N.OXIDE	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for oxide. **synonym: N.SIO2**	100	none
N.NITRID	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for nitride. **synonym: N.SI3N4**	100	none
N.OXYNIT	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for oxynitride.	100	none
N.SAPPHI	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for sapphire.	100	none
N.BPSG	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for BPSG.	100	none
N.INAS	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for InAs.	0.1	none
N.GAAS	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for GaAs.	0.1	none
N.ALGAAS	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for AlGaAs.	0.1	none
N.SIGE	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for SiGe.	0.1	none

Parameter	Type	Definition	Default	Units
N.HGCDTE	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for HgCdTe.	0.1	none
N.INGAAS	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for InGaAs.	0.1	none
N.SIC	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for SiC.	0.1	none
N.S.OXID	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for oxide when it is simulated as a wide band gap semiconductor.	0.1	none
N.GERMAN	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for Ge.	0.1	none
N.INP	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for InP.	0.1	none
N.DIAMON	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for diamond.	0.1	none
N.ZNSE	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for ZnSe.	0.1	none
N.ZNTE	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for ZnTe.	0.1	none
N.A-SILI	number	Mesh aspect ratio (normal spacing/lateral spacing) at boundaries for amorphous silicon.	0.1	none

Mesh Adjustments

Parameter	Type	Definition	Default	Units
PERIODIC	logical	Specifies that periodic boundary conditions are used in the horizontal direction. If this parameter is false, then reflection symmetry boundary conditions are used at the left and right edges of the device structure.	false	
PBC.TOL	number	The tolerance allowed between left and right edge nodes when determining whether a structure is valid for periodic boundary conditions.	0.1	fraction of local grid spacing
ORDER	logical	Specifies that the mesh nodes and mesh elements are ordered from left to right and from top to bottom.	True for structures created by Medici or if **PERIODIC** is specified; otherwise, false.	
ADJUST	logical	Specifies that the triangle diagonals are flipped whenever it improves (smooths) the shape of regions and electrode.	false	
VIRTUAL	logical	Generates virtual nodes at heterojunctions.	false	
CENTROID	logical	Specifies that, when an element produces a zero coupling term along an interface, a centroidal type discretization will be applied to the element. Use of this parameter normally results in a more stable discretization with reduced sensitivity to the mesh. A few cases have been observed where it is better not to use the centroids. These cases occur when Fowler-Nordheim or Band-to-Band currents are large, such as in some EEPROM devices.	true	
WIDTH	number	The width of a grid section that is inserted into the completed simulation structure at the x coordinate location specified with **X.SPLIT**. This parameter is particularly useful for varying the size of simulation structures that already exist and are read in using the **IN.FILE** parameter.	none	microns

Parameter	Type	Definition	Default	Units
N.SPACES	number	The number of grid spaces to use in the grid section that is inserted into the completed simulation structure.	none	none
X.SPLIT	number	The x coordinate location where an additional grid section is inserted into the completed simulation structure.	The midpoint of the structure.	microns
OBTUSE.A	number	The threshold angle that is considered obtuse. All elements with angles larger than the specified threshold angle will be reported as obtuse on the mesh statistics print-out and shown as obtuse on the plot. This parameter does not affect the way the elements are treated for the carrier continuity equations, where obtuse elements are those with negative coupling coefficients.	93	degrees
Mesh Output File				
OUT.FILE	char	The identifier for the output file to store the generated mesh. **synonym: OUTFILE**	none	none
ASCII.OU	logical	Specifies that the output mesh is stored in a formatted file. If this parameter is not specified, the output mesh is stored in a binary file.	false	
NO.TTINF	logical	Specifies that no triangle tree information is written to the output mesh.	false	
SMOOTH.K	number	Specifies a method for mesh smoothing. **SMOOTH.K**=1 indicates triangular smoothing is used, maintaining all region boundaries fixed. **SMOOTH.K**=2 indicates triangular smoothing is used, maintaining only material boundaries. **SMOOTH.K**=3 indicates that node averaging is used.	none	none

Description

A **MESH** statement can be used to either initiate the generation of a device structure or to read a previously generated device structure from a data file. It is possible to include several **MESH** statements in a single input file in order to perform simulations for multiple device structures. Whenever a **MESH** statement is encountered in an input file, Medici performs an initialization that allows a completely new simulation to be started.

See Also…

To further illustrate the **MESH** statement refer to input file *mdex1* in N-Channel MOSFET Examples, Chapter 4, "Mesh" on page 4-3.

Generating an Initial Mesh

This section describes how the **MESH** statement is used to generate an initial simulation structure.

Cartesian and Cylindrical Coordinates

When using the **MESH** statement to initiate the generation of a structure, you have the choice of the following coordinates:

- Cartesian
- Cylindrical

The default is to use Cartesian coordinates (**RECTANGU**). In this case, simulations are be performed in an xy-plane with the x-axis going from left to right and the y-axis going from top to bottom.

If cylindrical coordinates are chosen (**CYLINDRI**), the simulations are performed in an rz-plane. In this case, the Cartesian x-axis becomes the cylindrical r-axis and the Cartesian y axis becomes the cylindrical z-axis.

Statement Specification

Generating an initial simulation structure requires the specification of several statements in the proper order. Statement specification should be as follows:

1. Specify the **MESH** statement.
2. Specify the **X.MESH** and **Y.MESH** statements.

   These statements are used to define the initial placement of nodes.
3. There is now a choice between three sets of optional statements which can be used to modify the node placement.
   a. **ELIMINATE** statements may be used to eliminate unnecessary nodes and **SPREAD** statements may be used to distort the initial mesh.
   b. A **BOUNDARY** statement may be used to modify the initial mesh to conform to arbitrary boundaries.
   c. **TSUPREM4** statements may be used to define regions and profiles from the results of a TSUPREM-4 simulation.
4. Define the following for the structure:
   a. Material regions using the **REGION** statement.
   b. Electrode placement using the **ELECTRODE** statement.
   c. Impurity profiles using the **PROFILE** statement.
5. The generated mesh can be saved for a future simulation using the **OUT.FILE** parameter on the **MESH** statement, or by using the **SAVE** statement.
6. The initial mesh can be refined further, if desired, using the **REGRID** statement. In this case, the refined mesh should be saved using the **OUT.FILE** parameter on the **REGRID** statement, or by using a **SAVE** statement following the regrid.

Mesh Smoothing

The **SPREAD** statement has a tendency to create triangles with very obtuse angles which may lead to unphysical solutions or poor convergence. If the **SPREAD** statement is to be used in the mesh generation sequence, the **SMOOTH.K** parameter should be specified on the **MESH** statement.

SMOOTH.K=1 or 2 indicates triangle smoothing is used. Each adjoining pair of triangles is examined, and the diagonal of the quadrilateral is flipped if it improves the quality of the triangles. When two elements are of different materials, the diagonal is never flipped.

With elements of the same material but different region number, the diagonals are not flipped if **SMOOTH.K**=1 and may be flipped if **SMOOTH.K**=2. **SMOOTH.K**=3 indicates node smoothing, which repositions nodes in order to improve the angles of the triangles surrounding it.

Note:

Node smoothing is not recommended for a refined or distorted mesh since it tends to redistribute fine grid away from areas where the physical properties of the structure require it.

Periodic Boundary Conditions

Periodic boundary conditions in the horizontal direction are imposed if the parameter **PERIODIC** is specified. In this case, the left and right edges of the simulation region should be identical in terms of number of nodes, vertical location of nodes, doping, and electrode specifications.

If a different number of nodes exist on the left and right edges (defined as the minimum and maximum horizontal coordinates) or if the corresponding nodes are located at different vertical locations, an error message is issued. To ensure that the same doping is used on the left and right edges, the program automatically copies the doping from the left edge to the right edge.

Finally, if an electrode contacts one edge of the structure, but not the other, the program automatically converts the corresponding nodes on the other edge into electrode nodes, as well.

Note:

Periodic boundary conditions are not allowed in structures that are used in circuits when using the CA-AAM.

Automatic Boundary Conforming Mesh Generation

If a structure is read in from a TSUPREM-4, TIF, or Medici file using the **IN.FILE** parameter, a new mesh for the structure can be generated using the **ABC** parameter on the **MESH** statement.

If you specify the **ABC** parameter, the previous mesh from the imported structure is discarded and a new mesh is created using the automatic boundary conforming meshing algorithm. To provide better spacing control and improved robustness, a new version of the **ABC** mesh generator has been developed. You can still access the old version using the **ABC.OLD** parameter.

Algorithm

If a mesh is generated using **X.MESH** and **Y.MESH** statements, grid lines are aligned with the cartesian X and Y axes. Manual specification of the mesh spacings is required throughout the mesh. In some cases, the original geometry is distorted by shifting the location of the region boundaries to the nearest mesh nodes.

The automatic boundary conforming (ABC) meshing algorithm creates grid lines that conform to the region boundaries. The ABC algorithm also conserves the shape of the original regions and may be used without specifying meshing parameters. However, the ABC algorithm may fail to generate a mesh for a region with extremely complicated geometry.

The ABC mesh generator creates "layers" of grid that conform to the boundaries (both internal and external) of the original structure. The lateral grid spacing at the boundaries along these layers can be specified on a global basis with the **SPACING** parameter. In the new ABC mesh generator, non-uniform lateral spacing along the boundaries can be specified on a region-by-region basis using the **ABC.MESH** statement. The actual spacing along the boundaries may be smaller than the specified spacing in locations where the boundary is curved.

The normal grid spacing at the boundaries (thickness of the first grid layer) is specified as a ratio of the desired normal spacing to the value of the **SPACING** parameter using the **N.SEMICO**, **N.INSULA**, and **N.CONDUC** parameters for semiconductors, insulators, and conductors, respectively. The old version of ABC also allows specification of normal spacing ratios for other materials. The new version allows a more precise specification of normal spacing on a material or region basis and also allows the specification of non-uniform normal spacings through the **ABC.MESH** statement. The normal grid spacing away from the boundary increases toward the region's interior by a factor of **RATIO** or **NORMGROW** for each subsequent grid layer. The boundary conformal grid layers are created for materials that have values of the normal grid spacing much less than 1.0. For materials with the values of the normal grid spacing that are comparable or larger than 1.0, an unstructured mesh is often generated.

New Automatic Boundary Conforming Mesh Generator

The new version of the ABC mesh generator uses the same basic algorithm as the original ABC mesh generator. Layers of mesh elements are constructed to conform to the region boundaries according to spacing parameters provided by the user. The new version has been made more robust and, in particular, is capable of handling jagged boundaries and regions with multiple embedded regions. In addition, you can now specify spacing parameters on a local basis via the **ABC.MESH** statement. The **MESH** statement in conjunction with the **ABC** parameter and other associated parameters is used to initiate the ABC meshing process and to specify global meshing properties. For details on the new ABC mesh generator, please see the description of the **ABC.MESH** statement on page 3-40.

Spacing Parameters

The **MESH** statement in conjunction with the **ABC** parameter is used to specify global values of spacing parameters used for all regions and region boundaries. Some of the parameters used by the original ABC mesh generator are also used by the new ABC mesh generator. Many of the original parameters, however, are not applicable to the new ABC mesh generator, and a number of new parameters have been created to support additional functionality.

Gridding Region Boundaries

The ABC meshing of a device begins by re-gridding the boundaries between regions according to the spacing parameters specified by you. One key feature of this re-gridding process is how closely the newly gridded boundaries match the original boundaries. The deviation between an original and new boundary is controlled by using the **CRITICAL** parameter, which is an indication of the critical feature size that should be maintained. During boundary re-gridding, the new boundaries are guaranteed to deviate from the original boundaries by no more than the specified **CRITICAL** value. Specifying a small, but non-zero, value of **CRITICAL** allows the new boundaries to closely track the original boundaries while at the same time allows redundant nodes that are very close to each other to be eliminated. By default, **CRITICAL** is set at 1/1000 of the minimum of the device width and the device height.

The grid spacing used during the re-gridding of the region boundaries is specified on a global basis using the **SPACING** parameter, which by default is set at 1/50 of the device width. This produces a uniform spacing except along highly curved boundaries which are refined to satisfy the **CRITICAL** spacing parameter. More detailed control of the boundary gridding is obtained with the **ABC.MESH** statement which allows the specification of non-uniform gridding on a local basis.

All boundaries that are re-gridded generate layers during the meshing of the region interior. Boundary re-gridding is performed for all interior boundaries, and by default, the top boundary of the device. In some applications, however, it is desirable to leave the top ungridded and propagate mesh layers from the interior up to the top. Specify this by using the **GRIDTOP** parameter. The sides and bottom of the device are never gridded.

Gridding Region Interiors

The grid spacing into the regions, i.e. the normal spacing, beginning at the region boundaries is specified for general semiconductors, conductors, and insulators by using the ratio parameters **N.SEMICO**, **N.CONDUC**, and **N.INSULA**. These parameters specify the aspect ratio (normal spacing/boundary lateral spacing) of the elements near the region boundaries.The actual normal spacing that is used is given by the value of the **SPACING** parameter times the aspect ratio. By default, the semiconductor aspect ratio is set at 0.1, the conductor ratio at 0.2, and the insulator ratio at 1.0. The growth of the normal spacing into the region as layers are created is controlled by the **NORMGROW**, or **RATIO**, parameter which is set by default to 1.2.

During meshing of the regions, the lateral spacing is controlled using the **LATERAL** and **ELIMINA** parameters. The **LATERAL** parameter acts like a target value. Whenever the lateral spacing during meshing falls below **LATERAL**, the lateral spacing is unrefined. Likewise, if the lateral spacing during meshing rises above **LATERAL**, then the lateral spacing is refined. Refinement or unrefinement is only performed, however, if non-obtuse elements can be produced. Setting a very large value of **LATERAL**, or using the **ELIMIN** parameter, causes the mesh generator to unrefine the lateral spacing whenever non-obtuse elements are produced. By default, **LATERAL** is set at the maximum of the device width and device height.

Automatic Refinement Near Corners

To improve the mesh quality near corners where different regions meet, an automatic refinement algorithm has been developed to reconcile the different mesh spacings that often occur in corners. The **RFN.CRNR** causes the new ABC mesh generator to refine the lateral spacing along region boundaries to better match the normal spacing at the boundaries. For this to work effectively, however, it is necessary that the normal spacings in the different regions that meet at a corner be approximately the same.

Junctions

As in the original ABC mesh generator, use the **JUNC.ABC** parameter to conform grid lines to metallurgical junctions in addition to region boundaries. The position of the junction is determined from a logarithmic interpolation of the original doping profile.

Electrodes

The new ABC meshing generator correctly handles flat electrodes, including maintaining the original endpoints of the electrodes during the re-gridding of the region boundaries. Region electrodes can be voided using the **VOIDELEC** parameter which causes the region to be meshed without introducing any new nodes in the interior of the region. Both flat and volume electrodes from the original structure are maintained in the ABC-created mesh and do not need to be re-specified using the **ELECTRODE** statement.

Old Automatic Boundary Conforming Mesh Generator

The old ABC mesh generator uses the **MESH** statement to specify spacing parameters on a material-by-material basis. The **SPACING** parameter specifies a uniform lateral spacing along the region boundaries. The aspect ratio parameters such as **N.SILICO** can be used to specify the normal spacing at the region boundaries.

Lateral and Normal Mesh Spacings

As an example, suppose that a 0.25 micron MOSFET were meshed with the default set of parameter values. The structure width (the size of the simulation structure in the horizontal direction) for such a device is usually about one micron. By default, the spacing in silicon along the channel is 1/50 of the structure width (about 0.02 microns or 200 Å). The spacing in silicon in the normal direction to the oxide/silicon interface is a product of 200 Å and the mesh aspect ratio, defined for silicon by the parameter **N.SILICO**, which is 0.1 by default. For this example, 200 * 0.1 = 20 Å. Inside the silicon region from the oxide/silicon interface, each subsequent step size is larger than the previous one by the multiplication factor, defined by the parameter **RATIO**, which is 1.2 by default. In this example, the grid spacings in the channel in the vertical direction are 20 Å, 24 Å, 29 Å, 35Å, 41Å, 50 Å, ….

For regions with complex geometry, the grid layers are constructed along the entire boundary. For regions with straight left, right, and bottom sides, the grid layers are constructed from the top boundary only. Layer after layer, the boundary conforming elements are generated until the entire region volume is meshed.

Unstructured Delaunay Mesh

If it is not possible to mesh an entire region with a boundary conforming mesh, an unstructured Delaunay mesh generator is used for the residual subregion in the center of the region.

By default, the mesh aspect ratio (the ratio of the normal to lateral mesh spacings at the region boundary) is 0.1 for semiconductors and conductors, and 100 for insulators. This means that the elements along the boundaries of semiconductors and conductors are stretched, making the element aspect ratio about 10. The insulator aspect ratio of 100 usually requires that the unstructured Delaunay mesh generator be used for the entire region.

Selecting Aspect Ratios

The aspect ratios specified for the neighbor regions should be either similar to each other or very different, for example, 0.1 and 0.1, 0.2 and 0.2, or 0.1 and 100. When using the ABC algorithm, it is easier to generate non-obtuse elements when the aspect ratios in the neighbor regions are similar. If a large aspect ratio is specified for a region, the ABC algorithm is forced to mesh the entire region with the unstructured Delaunay mesh.

Automatic Failure Handling

If there are thin sections in a region, where too many nodes are placed in front of too few nodes, the Delaunay mesh generator may fail to mesh the region or a residual subregion. If such a failure occurs, the boundary conforming meshing algorithm automatically relaxes the meshing parameters to allow the Delaunay generator to mesh the region. The algorithm then attempts to generate the mesh until it is successful or reaches the allowed number of attempts. By default, the allowed number is four, which can be modified using the **ATTEMPTS** parameter.

Obtuse Elements

Usually, even along the curved boundaries, the boundary conforming meshing algorithm generates non-obtuse elements. Some obtuse elements may occur at sharp corners or in the center of the region. Statistics often show an increase in the number of obtuse elements when the ABC algorithm is used compared to the conventional mesh. However, most obtuse angles are of the order of only 90.00001 to 91 degrees and do not reduce the mesh quality.

Junction Conforming Mesh

If **JUNC.ABC** is specified the mesh generator will try to conform grid lines to metallurgical junctions that occur within the structure in addition to boundaries. The junction is determined by using linear interpolation from the doping at the nodes of the mesh that is read in. If this junction is very jagged, the ABC meshing algorithm may fail in its attempt to place a conforming grid around it. Boundary smoothing, which is invoked for values of **ANGLE** greater than zero, may help in this regard.

Terminal Handling

In the current version of the boundary conforming mesh generator, volume-less terminals are not supported. Only volume terminals are imported into simulation structures from the input file when the ABC parameter is specified. Electrodes can be also specified explicitly using the **ELECTRODE** statement.

Polysilicon is converted to electrode when reading an input file if **POLY.ELE** parameter is specified on the **MESH** statement. In such a case, the aspect ratio for polysilicon should be specified using the **N.ELECTR** parameter instead of the **N.POLYSI** parameter.

Large Geometry Features

If the simulation structure contains large geometry features, such as deep trenches, the simulation domain boundaries (the extreme left, right, and bottom edges of the structure) must be far enough from the feature to allow the ABC algorithm to mesh the entire device.

Previously Generated Meshes

This section describes how to read in previously generated meshes. It contains the following:

- Reading a previously generated mesh
- Restrictions and Limitations
- Compatibility

Read a previously generated two- or three-dimensional mesh by using the **INFILE** parameter.

Restrictions and Limitations

When a previously generated mesh is read, no additional processing is allowed using the **X.MESH**, **Y.MESH**, **ELIMINATE**, **SPREAD**, and **BOUNDARY** statements.

You may alter a previously read mesh in the following ways:

- Additional electrodes may be added to the structure using **ELECTRODE** statements.
- Regions may be redefined or modified using **REGION** statements
- Impurity profiles may be added using **PROFILE** statements.
- Additional grid refinement may be done by using the **REGRID** statement.
- Region shape can be smoothed by using the **ADJUST** parameter.

TSUPREM-4

A structure generated by TSUPREM-4 for input into Medici can be read by specifying the parameter **TSUPREM4** and using **IN.FILE** to identify the file where the structure is stored.

In addition to reading the mesh which was created by TSUPREM-4, the net and total impurity concentration at each node is read. By default, the entire structure is read. It may be truncated by specifying one or more of the parameters **X.MIN**, **X.MAX**, **Y.MIN**, or **Y.MAX**.

An electrode may be added to the bottom of the structure (which is determined by the maximum y coordinate of the structure read in) by specifying **ELEC.BOT**.

Other Programs and TIF

A structure generated by other programs for input into Medici can be read by specifying the parameter **TIF** and using **IN.FILE** to identify the file where the structure is stored. In addition to reading the mesh, the doping at each node is read.

A TIF file generated by Medici with the solution information saved in the TIF file, serves as an initial guess for further simulations. An electrode may be added to the bottom of the structure (which is determined by the maximum y coordinate of the structure read in) by specifying **ELEC.BOT**. The saved solution may also be used for plotting.

Smoothing Region Shape

This section describes how to improve the way the region and electrode shape is handled by using the **ADJUST** parameter.

The **ADJUST** parameter may be used when the mesh is generated using the **X.MESH** and **Y.MESH** statements.

If an **ADJUST** parameter is provided with a **MESH** statement, then the triangle diagonals at the region interfaces are flipped whenever it smooths the interface shape. A piece-wise linear region shape interpolation is used instead of the default staircase interpolation.

This parameter is especially useful if the region/electrode shape is non-rectangular (polygonal or circular). It is not recommended if strictly rectangular region/electrode shape is desired, because it smooths both “convex” and “concave” corners of the rectangular region/ electrode.