

Simulation Standard

Connecting TCAD To Tapeout

A Journal for Process and Device Engineers

Simulating Accurate 3D Geometries for Interconnect Parasitic Extraction Using *CLEVER*

Introduction

CLEVER is designed to model interconnect parasitics by simulating the back end processing steps of custom cells in three dimensions. A 3D process simulator is used to realistically reproduce the effects of photolithography, deposition and etching on the resulting structure topography. Geometric etches and depositions are also possible to increase the speed of simulation if required. The process steps are driven from existing masks in standard GDSII data format from the actual design, or the mask set may be created using *MaskViews*. The 3D grid re-meshing required for each of the steps is generated totally automatically and requires no user input. This feature makes the whole package exceptionally easy to use, even for those unfamiliar with process simulation. Figure 1 shows an SRAM cell geometry during processing. Effects of lithography on the metal 1 layer are seen.

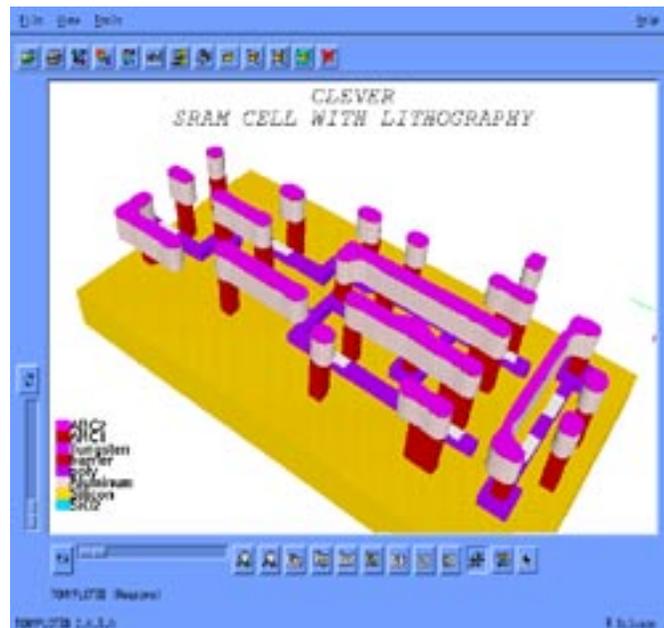


Figure 1. 3D selection of an SRAM cell after metal 1 lithography and etch. Accurate description of the as-processed geometry is required for correct parasitic extraction.

Interconnect Analysis

Once all the process steps have been simulated, the addition of the following single line to the input file:

```
Interconnect Capacitance Adapt
```

will initiate the integral 3D field solver in *Clever* to calculate the capacitance matrix between each conductor and every other conductor and between each conductor and the substrate. This will create an accurate netlist of the parasitic capacitive loading effect on each node for

SPICE circuit simulation. The analogous command:

```
Interconnect Resistance Adapt
```

will calculate a SPICE netlist of the parasitic resistance of each of the tracks between each node. The effects of

Continued on page 2....

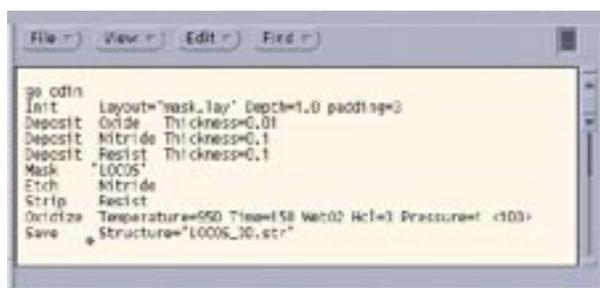


Figure 2: An Example Input File to Create a 3D LOCOS Structure

INSIDE

<i>Fourier Transform Analysis for Large Signal Frequency Response in ATLAS</i>	5
<i>Simulating Redeposition During Etch Using a Monte Carlo Plasma Etch Model</i>	6
<i>Calendar of Events</i>	9
<i>Hints, Tips, and Solutions</i>	10

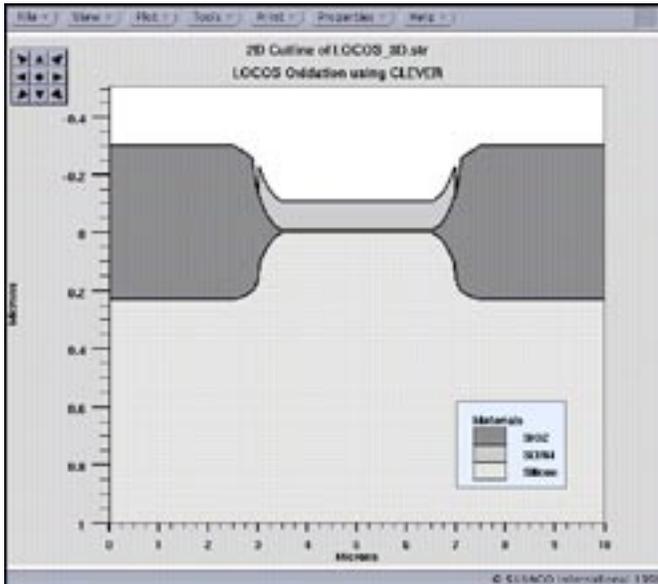


Figure 3. Showing a cutline through the 3D LOCOS structure generated in *Clever*.

using realistic 3D process simulation on the electrical timing characteristics of logic cells has been discussed previously, in the June 1997 issue of *Simulation Standard*. In this article, we focus on some of the 3D processing capabilities of *Clever* that are more advanced and were therefore not described before. Here the ease of creating an realistic input file is also emphasized.

Simulating Local Oxidation

Although not strictly considered as “back end” processing, the capability to simulate a standard LOCOS structure available. This example is shown to demonstrate the ease of use of the program and how it closely resembles a real process flow. The complete input file shown in Figure 2 is all that is required to create a LOCOS structure example using Silvaco’s 3D process simulator.

A description of the input file is as follows: Firstly *Odin* the 3D process simulator used by *Clever* is called by the “go odin” statement. Next the group of layout masks called “mask.lay” is used to initiate the size of the starting substrate. The “Depth” parameter defines the thickness of the substrate and the “padding” parameter tells the simulator to increase the area simulated by an additional 3um around the edge of the masks. Incidentally, all references to thickness are in microns.

The following “Deposit”, “Etch” and “Strip” statements are self explanatory. The “Mask” statement tells the simulator to deposit photo resist, and develop a pattern defined by the mask called “LOCOS”. After etching

the nitride, stripping the resist and oxidizing, the 3D structure is then saved in a structure file called “LOCOS_3D.str” and can be plotted using the 3D graphics package *TonyPlot3d*.

Notice that there is no reference at any time to the creation or definition of the three dimensional grid required for the process simulation. All of this is taken care of and is invisible to the user. The user can, of course, plot the grid that has been created, as well as any of the other parameters, using *TonyPlot3d*.

Defining Two Dimensional Cutlines

Although designed as a 3D simulator, *Clever* can be used in 2D mode (or even 1D mode) by defining a cutline through the mask set. All the advantages of automatic grid definition and ease of input syntax are retained if the process is run in two dimensional mode. Obviously the bonus of 2D simulation is a huge increase in simulation speed.

Using the input file in figure 2 as an example, if the syntax below is added to the end of the input file above, the simulator makes a cutline through the mask set at a location defined (X1, Y1, X2, Y2), saves a structure file and plots the result using the 2D graphics package, *TonyPlot*.

```
Cut line=(0, 5, 10, 5)
Save Structure="LOCOS_2D.str"
tonyplot LOCOS_2D.str
```

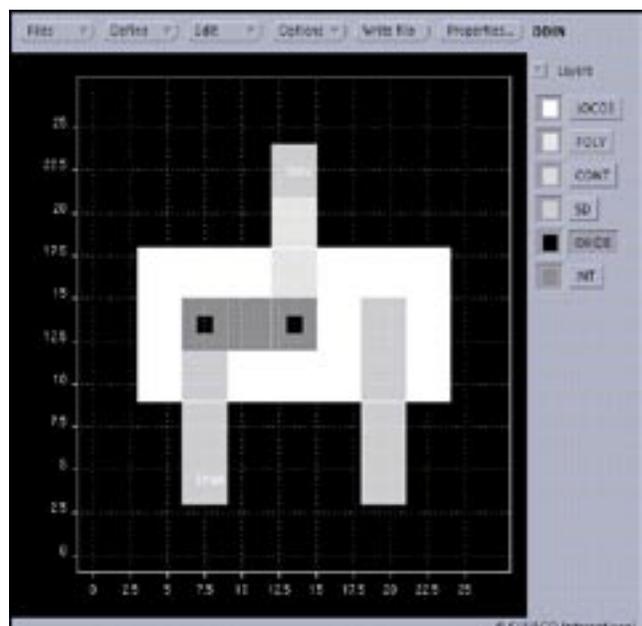


Figure 4. Showing the masks made using *MaskViews* required to generate the 3D Local-Interconnect structure.

```

File View Edit Find
go odin
Init Structure = "LOCOS_3D.str"
Deposit Poly rate=1.00 time=0.20 DT=0.02
Mask "POLY"
Etch PolySilicon
Strip Resist
Deposit Oxide rate=1.00 time=0.04
Mask "OXIDE" Reverse
Etch Oxide
Strip Resist
Deposit Material("TiN") rate=1.00 time=0.05
Mask "TiN"
Etch Material("TiN")
Strip Resist
Deposit BPSG rate=1.00 time=0.7
Mask "CONT" Reverse
Etch BPSG
Etch Oxide
Strip Resist
Deposit Aluminum rate=1.00 time=0.4
Mask "S0"
Etch Aluminum
Strip Resist
Electrodes "S0" Aluminum
Material Material("TiN") Conductivity=187000
Interconnect Resistance Adapt
Save Splice="cloud4.net"

```

Figure 5. Example Input File to Create a MOSFET with Local Interconnect

Once again, notice the ease of creating the input file and the intuitive command language that is used. The result of this simulation is shown in Figure 3.

3D Modeling of Local Interconnect

Now that a LOCOS structure is created, the processing sequence may be continued to create a further example structure with a local interconnect. This allows the study of the parasitic local interconnect resistance variations with different geometries and thicknesses. The mask layers for the various process steps were created using *MaskViews* and are shown in Figure 4.

The simulation begins by loading in the 3D LOCOS structure that was created in the previous example, depositing poly, then a thin layer of oxide, etching contact holes for the local interconnect and then depositing and defining the local interconnect material. This is followed by traditional back end processing consisting of bpsg deposition, etching and aluminum deposition. The example input file is shown in Figure 5.

In this part of the input file, instead of simple deposit statements, the physical deposit model is automatically called by specifying a deposit rate and time. Realistic photolithography and etching models can also be called by using similar syntax in the input file for these processes. The final structure is shown in Figure 6 with the oxide layer removed for clarity. A 2D outline of the same structure near the local interconnect is shown in Figure 7.

A further feature demonstrated in this input file is the "Reverse" parameter in the "Mask" statement. The "Reverse" parameter reverses a mask from dark field to light field or vice versa. In processing terms, this has the same effect as using negative resist as opposed to the default positive resist. This feature is useful when designing masks for certain processes such as via etches in the oxide, where it is easier and more useful to draw and visualize where the contact holes are, rather than draw a mostly dark field mask.

At the end of the process simulation, the conductivity of the TiN interconnect material is defined. This conductivity number should be extracted from test structures on the wafer for an individual process. The "Interconnect Resistance Adapt" command calculates the resistance of all conducting tracks between each node and forms a resistance matrix for inclusion in a file as a SPICE netlist. This is saved as a file in the last line of the input file.

Using the Simple Deposit Statement to Simulate a Damascene Structure

The final example shows how a simple deposit statement, using one of the geometric model options, can simulate a deposition followed by Chemical Mechanical Polishing (CMP). It also demonstrates performing a whole process simulation in two dimensions which takes only a few seconds of run time. The input file is shown in Figure 8.

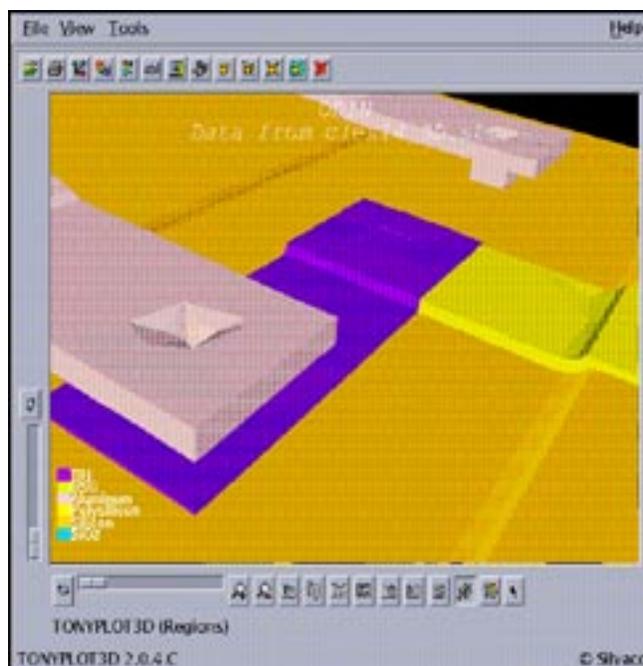


Figure 6. Graphics from TonyPlot3D showing the local interconnect bridging the polygate and service contacts.

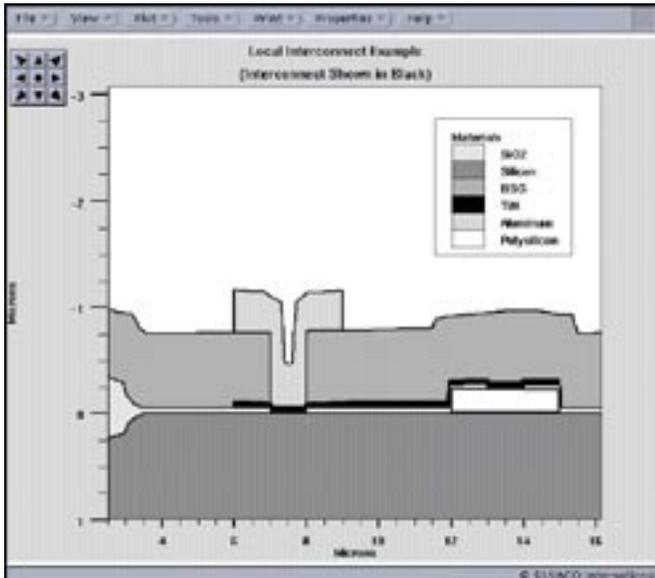


Figure 7. 2D cutline generated by **TonyPlot3D** showing section through the 3D structure near the local interconnect.

The meaning of the initial statements has already been described. For geometrical depositions, there are three options, namely "Max", "Min" and "Conformal", the default setting being "Max". If the parameter "Max" is added to the "Deposit" statement, the deposited thickness will be measured from the highest point on the present structure, and the deposit will fill all voids below this height. This has the effect of deposition and planarization in one deposit statement. The effect is shown in Figure 8, where it can be seen that "LoK" material appears as a deposited and polished (planarized) film.

The other geometrical options on the deposit statement are "Min" and "Conformal". The "Min" statement is similar to the "Max" statement except that the deposited thickness is measured from the lowest height of any part of the structure. Conformal is the more typical deposit statement and deposits a uniform thickness of material which follows the contours of the structure.

After the structure has been simulated, the "Interconnect Capacitance Adapt" command calculates the capacitance matrix which is then saved as a SPICE netlist.

Conclusion

Various 3D process simulation features in Clever have been demonstrated with actual input files. These include the formation of LOCOS and local interconnect structures together with the three versatile options available using geometrical deposition. It has been shown that the cutline feature allows the 3D process simulation to be reduced to a 2D simulation at any point in the simulation. The ease of using physical models has

```

go edin
Init Layout="LoK Lay" Depth=1.0 Padding=3.0
Cut Line=Ct, 6.75, 11, 6.75)

Deposit *Oxide Thickness=0.5
Deposit *Aluminum Thickness=0.8
Mask *MT1
Etch Aluminum
Strip Resist
Electrodes *MT1 Aluminum
Deposit Oxide Rate=.00 Time=0.05
Deposit Material("LoK") Thickness=0.05 Max
Deposit Oxide Thickness=0.5

Save Structure="LoK.str"
toyplot
LoK.str

Material Material("LoK") Permittivity=2.0
Interconnect Capacitance Adapt
Save SCICE="LoK.net"

```

Figure 8. Low k dielectric in-fill Damascene type structure generated using simple process commands.

also been demonstrated. The physical models available include photoresist development, film deposition and etching. An oxide growth model is also incorporated into the code.

The examples show how to define material properties such as resistivity and permittivity and how to implement the resistance and capacitance matrices calculation for inclusion into a SPICE netlist for circuit simulation. These input files are designed to show the ease of use of the program, in particular, the complete absence of any mesh definition requirement as a result of the automatic 3D mesh generation and adaption facility.

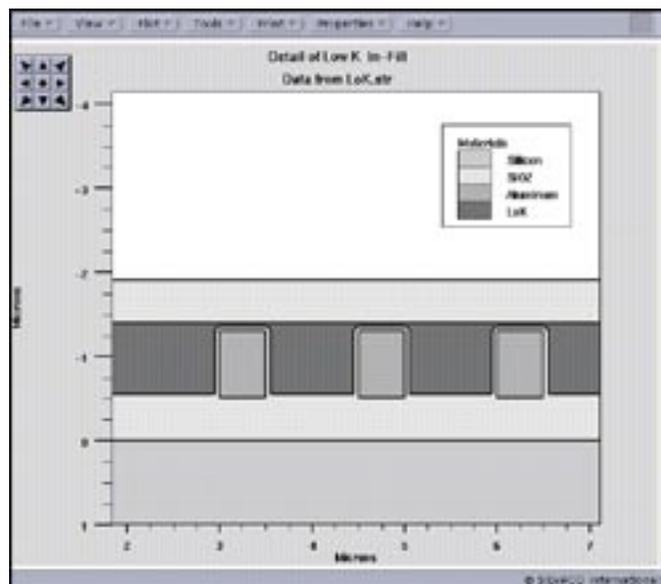


Figure 9. 2D cross-section of a low k dielectric fill process. Correct handling of the inter metal geometry is required to obtain accurate lateral capacitance.

Fourier Transform Analysis for Large Signal Frequency Response in *ATLAS*

The *ATLAS* device simulation framework has been to improve the analysis the large signal transient behavior. These features include a simple specification for single frequency sinusoidal waveforms and frequency domain representation of large signal data. Applications include all classes of RF devices in both silicon and III-V technologies.

A single frequency sinusoid can be specified on the *SOLVE* statement by using the *TRANS.ANALY* and the *FREQUENCY* parameters. The number of periods can then be set by using the *CYCLES* parameter.

The data stored in a *LOG* file from a transient simulation can be transformed from the time domain to the frequency domain by using the new *FOURIER* statement. This performs a Fast Fourier Transform (FFT) on the *LOG* file data, calculating the magnitude and phase spectra of each entry, in addition the real and imaginary values.

The syntax of the *FOURIER* statement is as follows:

```
FOURIER  INFILE  OUTFILE  [T.START  
T.STOP  FUNDAMENTAL  MAX.HARMONIC  
NUM.SAMPLES          INTERPOLATE  
REAL.VALUES]
```

The data contained in the *LOG* file specified by *INFILE* will be transformed and written to *OUTFILE*. The user can select the time window to be transformed by use of the *T.START* and *T.STOP* parameters. By default *FOURIER* will use all of the transient simulation data.

The fundamental frequency will be automatically calculated, but can be overridden by specifying *FUNDAMENTAL* on the *FOURIER* statement. The maximum harmonic frequency calculated by the FFT is determined by the number of discrete samples (*NUM.SAMPLES*) and the fundamental frequency. The number of samples should be 2^n where n is a positive integer. Alternatively, the maximum harmonic frequency can be set by using the *MAX.HARMONIC* parameter. In this case, the number of samples is determined by *ATLAS*.

The FFT requires uniformly spaced data samples from the transient simulation. Typically in *ATLAS* however the time steps are not uniformly spaced. *INTERPOLATE* must be specified on the *FOURIER* statement. *ATLAS* will then interpolate the *LOG* file data, ensuring that the time steps are uniform within the FFT.

The real and imaginary values can be stored in the output file by specifying *REAL.VALUES*.

Application Example

To illustrate the use of the FFT, a 0.1GHz sine wave was applied to the anode of a silicon diode. Figure 2 shows the resulting nonlinear anode current response. Figure 1 shows the magnitude of the anode current after it has been transformed into the frequency domain. To increase the frequency resolution, 10 periods of data were used by the FFT. It can be seen that additional harmonic frequencies are present at 0.2, 0.3, 0.4, 0.5 and 0.6GHz.

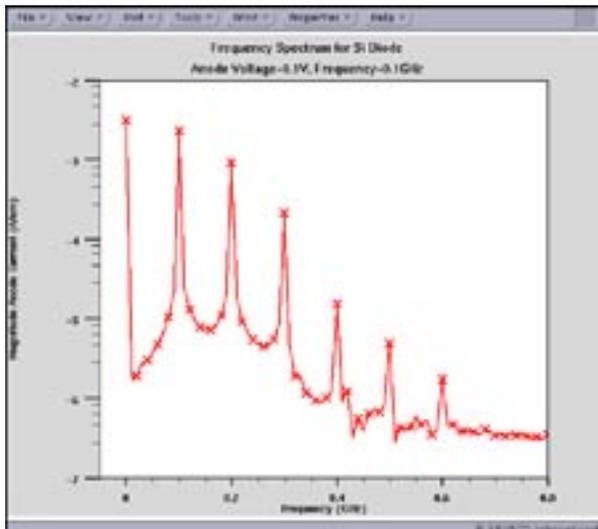


Figure 1. Large signal frequency spectrum exacted from *FOURIER* analysis.

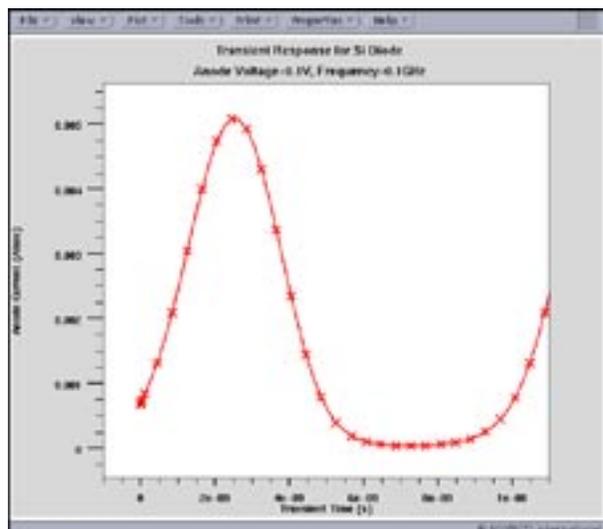


Figure 2. Non linear large signal response of silicon diode.

Simulating Redeposition During Etch Using a Monte Carlo Plasma Etch Model

Introduction

The shrinking critical dimensions of modern technology place a heavy requirement on optimizing the etching of narrow mask opening. In addition the aspect ratio of etches has been increased requiring deeper etches along with the small CDs. The simulation of these process requires more advanced techniques than the directional rate-based etching found in the current versions of *Elite*. A more complete treatment involving calculation of the plasma distribution is required.

The new Monte Carlo etch module is implemented into *ATHENA/Elite*. The main application of the module is simulation of plasma or ion assisted etching. The module can take into account the redeposition of the polymer material generated as a mixture of incoming ions with etched (sputtered) molecules of substrate material. In addition, the module has interface to the C interpreter which allows simulation of several other processes like wet etch and deposition, ion milling and sputtering deposition of various materials.

Simulation of Incoming Ions and Neutrals

Direct modeling of the plasma sheath is not included into this release and will be added later. It is assumed that ions and neutrals fluxes leaving plasma sheath are represented by bimaxwell velocity distribution function along the direction determined by user specified incident angle:

$$f(v_{\parallel}, v_{\perp}) \sim I \cdot \exp\left(-\frac{v_{\parallel}^2}{T_{\parallel}} - \frac{v_{\perp}^2}{T_{\perp}}\right) \quad \text{Eq. 1}$$

where v_{\parallel} is the ion velocity component parallel to the incident direction, v_{\perp} is the ion velocity component perpendicular to the incident direction, I ion (or neutral) current density, T_{\parallel} is the dimensionless parallel temperature and T_{\perp} is the dimensionless lateral temperature.

Calculation of Ion and Neutral Fluxes

During each time step the simulation consists of the three stages:

1. calculation of ion, neutral, and polymer fluxes
2. calculation of etch, polymer ejection and redeposition rates
3. surface movement

On the first stage, the fluxes of incoming and reflected ions and neutrals are calculated on the each segment of the surface. Computation of the ion fluxes is done by tracing of user defined number of particles (Figure 1a). Each particle is generated at the random position on the top of the simulation area with normal and lateral velocities randomly determined from the bimaxwell distribution function (Eq. 1). Then each particle trajectory is

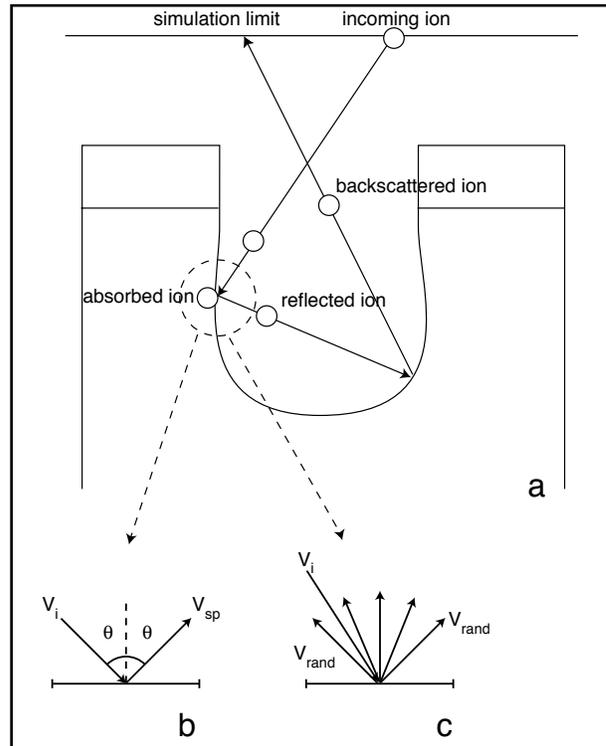


Figure 1. Diagram of Plasma Flux algorithm (a) including zoom-in of ion reflection models (b and c).

traced until the ion is either absorbed by the surface or back scattered out of the simulation area.

Interaction of the ion with material surface is governed by two parameters: reflection coefficient P_{refl} and roughness of the surface R both of which depend on the surface material and the type of ion. Reflection coefficient is the probability of the particle to be reflected from the surface. Roughness determines how the ion is reflected. If $R = 0$ the reflection is specular (Figure 1b), if $R = 1$ the reflection is random with uniform angular distribution (Figure 1c). In general case, the velocity of the ion after a collision with a surface segment could be presented as follows:

$$\mathbf{V}_{refl} = 0 \text{ (ion is absorbed), if } x > P_{refl} \quad \text{Eq 2a}$$

$$\mathbf{V}_{refl} = \mathbf{V}_{sp} \cdot (1 - R) + \mathbf{V}_{rand} \cdot R, \text{ if } X < P_{refl} \quad \text{Eq. 2b}$$

Where X is random number, and $|\mathbf{V}_{sp}| = |\mathbf{V}_{refl}| = |\mathbf{V}_i|$

Each absorbed ion contributes to the incoming flux F_i at the surface segment. The following characteristics describe the flux:

- normalized number of absorbed particles N_i

$$N_{norm} = N_{abs} \cdot I / N_{traj} \quad \text{Eq 3}$$

where N_{abs} number of absorbed particles, I is current density for the given type of the incident ion, N_{traj} number of trajectories;

- normalized normal and tangential velocity components of the absorbed particle before the encounter with the surface:

$$V_{\perp \text{abs}} = \frac{I}{N_{\text{traj}}} \cdot \frac{\sum V_{\perp}}{N_{\text{abs}}} \quad \text{Eq. 4a}$$

$$V_{\parallel \text{abs}} = \frac{I}{N_{\text{traj}}} \cdot \frac{\sum V_{\parallel}}{N_{\text{abs}}} \quad \text{Eq. 4b}$$

- normalized kinetic energy of absorbed particles:

$$V_{\text{abs}}^2 = \frac{I}{N_{\text{traj}}} \cdot \frac{\sum v^2}{N_{\text{abs}}} \quad \text{Eq. 5}$$

Calculation of Polymer Fluxes

After ion and neutral fluxes are determined, the fluxes of the polymer particles are calculated as follows. As the result of ion flux interaction with the surface segment the polymer particles are generated. The angular distribution of the polymer particles is uniform and the current density of these particles is determined by the etch model (see below) and the sum of the fluxes from incoming ions, neutrals, and from polymer particles ejected from other surface segments. Obviously, the latter flux needs to be pre-calculated.

This flux is computed as follows. First, the configuration (or geometrical) factors are calculated. These factors are the fractions of the number of particles ejected from one segment and absorbed by the other one. These are calculated using the same trajectory tracing algorithms which are described above for the incident ions and neutrals with the only one difference that starting points are not at the upper boundary of the simulation area but at the surface segments. After this an iteration process is initialized. At the first iteration only the incoming ion and neutral fluxes are used for calculation of the ejection rates from each surface segment. Knowing the current densities of ejected

particles and the configuration factors the polymer fluxes are calculated. At the subsequent iterations the polymer fluxes calculated at the previous iteration are used to update the etch and ejection rates. The iterations are repeated until etch and ejection rates converge.

Calculation of Rates

The second stage involves calculation of the etching rates as well as ejection and redeposition rates of the polymer particles. During each time step the two processes simultaneously take place on each surface segment. The first is redeposition of the polymer with the rate equal to the polymer flux. The second is etching by incoming ions and neutrals. The combination of these two processes can be treated as deposition of a virtual polymer layer with subsequent etching of the two-layer structure. If the etch rate of polymer by incoming ions and neutrals is less than the polymer deposition rate the result is redeposition of a polymer layer on the surface. If the etch rate of polymer by incoming ions and neutrals is larger than the polymer deposition rate the result is actual etch of the underlying material. In the case of the linear model the etch rate is calculated as:

$$\text{EtchRate (mat)} = \sum_{\text{ion types}} \text{EtchParam (mat, ion)} \cdot V_{\text{abs}} \quad \text{Eq. 6}$$

If EtchRate (polymer) is less than the polymer flux (redeposition rate) the actual etch rate is negative which corresponds to redeposition:

$$\text{EtchRate} = \text{EtchRate (polymer)} - \text{Polymer Flux} < 0 \quad \text{Eq. 7}$$

$$\text{Ejection Rate} = \text{Etch Rate (polymer)} \quad \text{Eq. 8}$$

EtchRate(polymer) is larger than polymer flux the actual etch rate is positive:

$$\text{EtchRate} = \text{Etch Rate (mat)} - \text{Polymer Flux} \sum_{\text{ion types}} = \frac{\text{Etch Param (mat, ion)}}{\text{Etch Param (polymer, ion)}} \quad \text{Eq. 9}$$

$$\text{Ejection Rate} = \text{Polymer Flux} + \text{Etch Rate} \quad \text{Eq. 10}$$

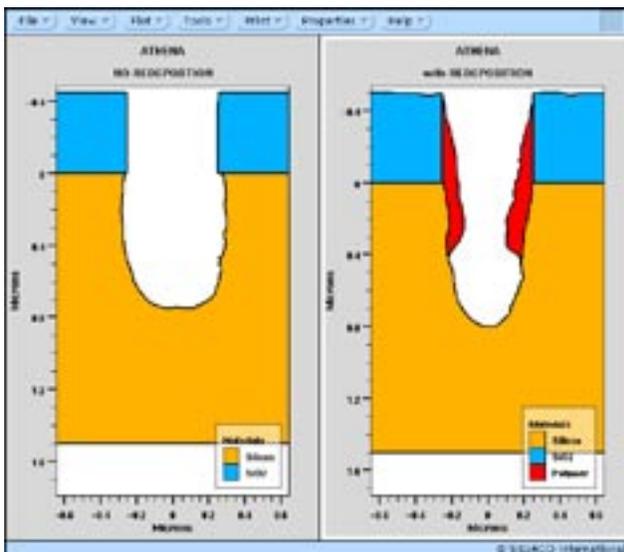


Figure 2. Comparison of Silicon trench etch with and without polymer redeposition.

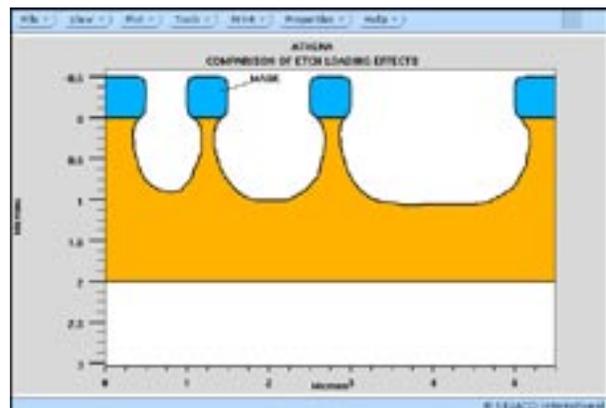


Figure 3. Demonstration of loading effects using the MC plasma etch model. No redeposition was considered here.

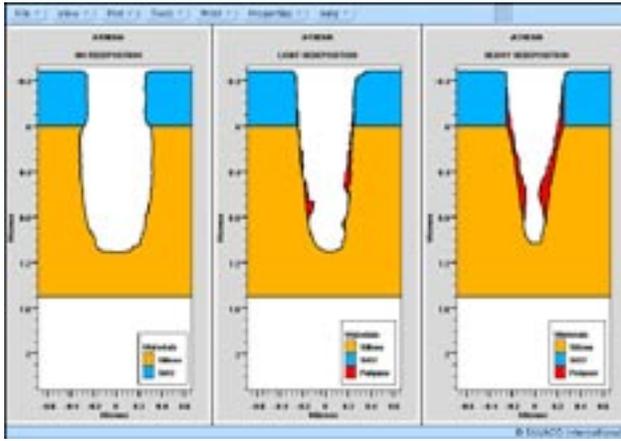


Figure 4. User control over the redeposition parameters is important for tuning. Redeposition converts a 'barrel' shaped trench into a straight sidewall trench.

C-interpreter

C-interpreter can be used for introduction of different etch and ejection models. The following parameters are passed to the C-interpreter file and can be used for implementing the models: number of ion types, the four characteristics of ion fluxes for each ion type (Eq 3 - Eq 5), PolymerFlux, and surface material. Returned parameters are EtchRate and EjectionRate.

For example, the wet etching can be simulated by setting the etch rate to a constant positive value depending only on the surface material. In this case the trajectory tracing part of the model is not needed, the number of trajectories can be set to one.

Uniform deposition can be simulated by setting of negative constant etch rate and specifying the redeposited material other than polymer in the etch statement. If the fluxes are not used as in the wet etching simulation the void formed will be eventually filled with the deposited material because inside the C-interpreter there is no way to determine if the current surface segment belongs to the void or not. This obstacle can be overcome by simulating ion fluxes and setting the etch rate equal to zero if the flux on the surface segment is less than some small threshold value.

Surface Movement

A sophisticated string algorithm is used to move all segments according to the rates (positive or negative) calculated at each time step. If the rate is negative the surface moves outside and the area is filled with redeposited material (by default, polymer). If the rate is positive the surface moves inwards and the area is filled with vacuum.

Applications

This model can be used to simulate the redeposition of material during etches for:

- deep and narrow trench etches
- via etches
- loading effects

Figure 2 shows the etch of a deep narrow trench in silicon. It shows a comparison between the case with no redeposited material and the case including redeposition. Redeposition can be defined as selective between the silicon, mask and other materials. The redeposited materials continually etched during subsequent timesteps in the etch simulation according to the plasma flux.

Figure 3 shows the loading effects during etch where the depth of the etch depends on the size of the mask opening. The reflection parameter R can be used to tune this effect.

The physics of the polymer redeposition can be controlled by the user. Figure 4 illustrates the effect of changing the polymer redeposition rate on the final etch profile. The case with no redeposition produces a 'barrel' shaped etch. The polymer acts to passivate the sidewalls of the trench, reducing the lateral etch rate. This leads to trenches with more vertical sidewalls as shown in experimental work such as [1].

Figure 5 shows the effect of mask size opening on the etch characteristics when polymer re-deposition is considered. Both the 0.25 micron and 0.5 micron mask openings are processed with the same etch conditions. The polymer redeposition in the 0.25 micron case inhibits the etch rate at the bottom of the etched area. The same redeposition physics applied to a larger mask opening shows a larger etch rate as the polymer is unable to inhibit the plasma in the center of the mask opening.

Summary

A new model for Monte Carlo plasma etching has been implemented into *ATHENA*. It allows physically based modeling of the mechanism of polymer deposition during etches. The application of this model to aid process optimization of deep narrow trench or via etches.

Bibliography

- [1] "Simulation Study of Micro-Loading Phenomena in Silicon Dioxide Hole Etching", Misaka and Harafuji, IEEE Electron Devices May 1997.

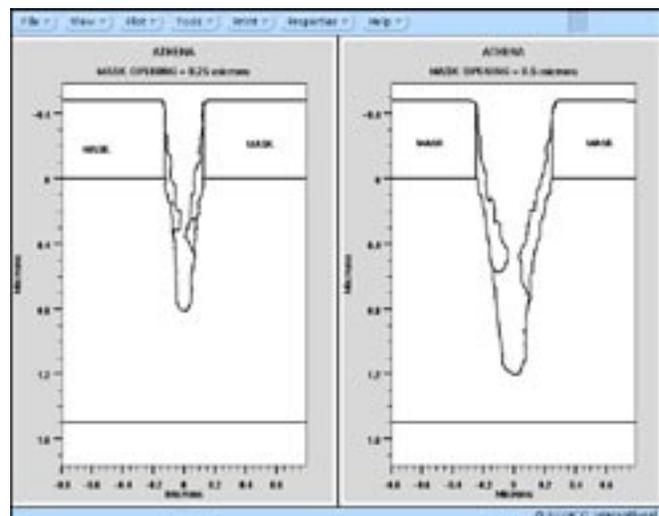


Figure 5. Etch depth varies with the size of the mask opening as the redeposited material restricts etching the bottom of the trench.

Calendar of Events

August

1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20 W/S - San Diego
21 W/S - San Diego
22
23
24
25
26
27
28
29
30
31

September

1
2 SISPAD - Belgium
3 SISPAD - Belgium
4 SISPAD - Belgium
5
6
7 ESSDERC '98 - France
8 ESSDERC '98 - France
9 ESSDERC '98 - France
10 ESSDERC '98 - France
11 ESSDERC '98 - France
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27 BCTM Conference
28 BCTM Conference
29 BCTM Conference
30

Bulletin Board



Silvaco Presentation at LCD/PDP International Seminar

Silvaco's Director of Application Engineering, Andy Strachan, will present a talk on "TCAD Driven CAD for TFT Devices" at the LCD/PDP Conference in Makuhari Messe, Japan on 29th October 1998. The presentation will cover an integrated process, device and circuit modeling solution for polysilicon thin film transistor technology.



SOI Conference Exhibition

Silvaco will be presenting the latest advances in simulation and modeling of SOI technologies at the IEEE International SOI Conference in Stuart, Florida on 5th-8th October. Silvaco engineers will be demonstrating circuit simulation, SPICE model extraction and TCAD tools. Highlights include TCAD verified circuit simulation capacitance models and oxide reliability simulation.



Exhibition of 3D Interconnect Parasitic Extraction at ESSDERC '98

The latest version of *Clever* will be presented in Europe at the ESSDERC '98 conference in Bordeaux, France from 7th to 11th September. Staff from Silvaco's three European offices will be on-hand to demonstrate the most accurate approach to interconnect parasitic extraction using 3D process simulation.

For more information on any of our workshops, please check our web site at <http://www.silvaco.com>

The Simulation Standard, circulation 17,000 Vol. 9, No. 8, August 1998 is copyrighted by Silvaco International. If you, or someone you know wants a subscription to this free publication, please call (408) 567-1000 (USA), (44) (1483) 401-800 (UK), (81)(45) 341-7220 (Japan), or your nearest Silvaco distributor.

Simulation Standard, TCAD Driven CAD, Virtual Wafer Fab, Analog Alliance, Legacy, ATHENA, ATLAS, FastATLAS, ODIN, VYPER, CRUSADE, RESILIENCE, DISCOVERY, CELEBRITY, Manufacturing Tools, Automation Tools, Interactive Tools, TonyPlot, DeckBuild, DevEdit, Interpreter, ATHENA Interpreter, ATLAS Interpreter, Circuit Optimizer, MaskViews, PSTATS, SSuprem3, SSuprem4, Elite, Optolith, Flash, Silicides, SPDB, CMP, MC Deposit, MC Implant, Process Adaptive Meshing, S-Pisces, Blaze, Device 3D, Thermal 3D, Interconnect 3D, Blaze3D, Giga3D, MixedMode3D, TFT3D, Luminous3D, TFT, Luminous, Giga, MixedMode, ESD, Laser, Orchid, Orchid3D, SiC, FastBlaze, FastMixedMode, FastGiga, FastNoise, MOCASIM, UTMOST, UTMOST II, UTMOST III, UTMOST IV, PROMOST, SPAYN, SmartSpice, MixSim, Twister, FastSpice, SmartLib, SDDL, EXACT, CLEVER, STELLAR, HIPEX, Scholar, SIREN, ESCORT, STARLET, Expert, Savage, Scout, Dragon, Maverick, Guardian and Envoy are trademarks of Silvaco International.

Hints, Tips and Solutions

Andy Strachan, Applications and Support Manager

Q: How can the lateral implant straggle be tuned separately to the vertical implant profile?

When using the analytical implant tables in *ATHENA* the default model for the lateral implant range assumes a gaussian profile with a standard deviation equal to the vertical standard deviation (ΔR_p) read from the table. This approximation is reasonable for amorphous implant substrates. This includes cases where the implant beam has been amorphised by a screen oxide or is implanted at a high angle.

Tuning of the lateral implant range can be done by specifying the lateral standard deviation prior to the implant:

```
MOMENTS ..... LSTD.DEV=<value>
IMPLANT .....
```

However this is cumbersome to use in many cases since the `MOMENTS` statement requires explicit definition of the implant dose and energy and all the vertical implant parameters. Therefore it is more convenient to use a unit less scaling parameter `LAT.RATIO1` on the `IMPLANT` statement to multiply the default lateral straggle by a factor. The default for `LAT.RATIO1` is 1.0 so to slightly increase the lateral straggle of an arsenic implant the following syntax might be used:

```
IMPLANT ARSENIC DOSE=<v> ENERGY=<v> \
LAT.RATIO1=1.2 PRINT.MOM
```

This `PRINT.MOM` parameter is included as a useful check. It prints the implant moments used into the run-time

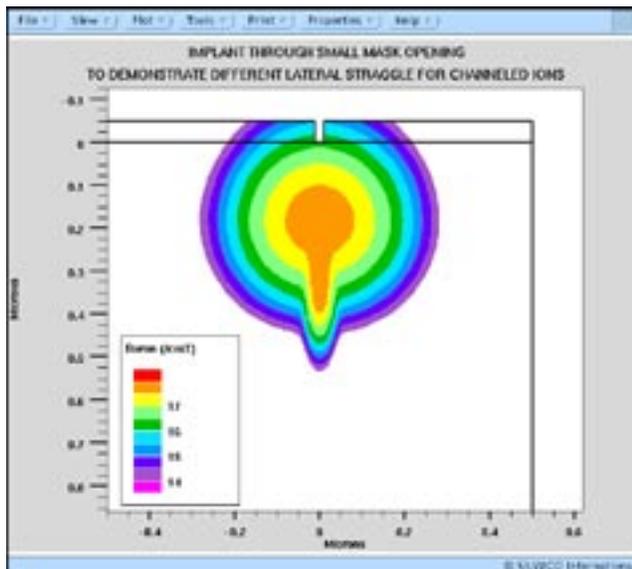


Figure 2. Demonstration of different lateral straggle of channeled and non-channeled ions.

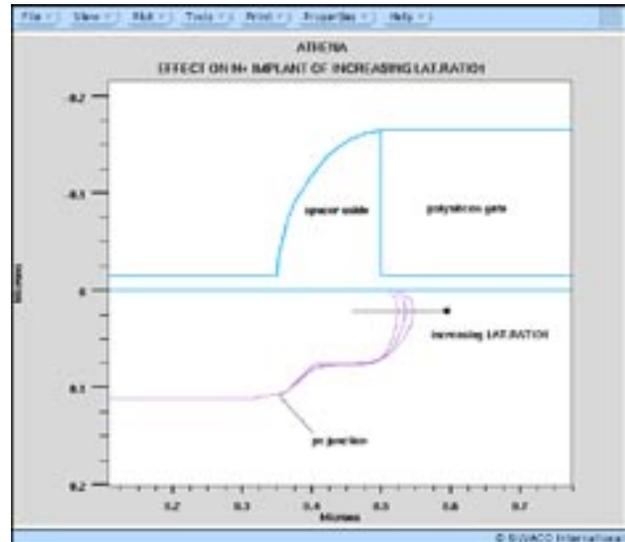


Figure 1. Using the `LAT.RATIO1` parameter to tune lateral implant range.

output. Figure 1 shows the effect of adjusting only `LAT.RATIO1`. The vertical implant depth is constant while the lateral implant straggle can be tuned to match electrical results.

For implants that are normally modeled with a dual Pearson distribution (eg. the SVDP Model) a separate lateral straggle is used for each of the two Pearson distributions. This means for an implant using the SVDP Model there is a separate lateral straggle for the main and secondary dopant distributions. Since the secondary distribution in the Dual Pearson is from ion channeled through the lattice, the lateral straggle of these ions is much reduced. By default it is equivalent to 0.2 times the standard deviation of the secondary Pearson distribution (`SSTD.DEV`). For most implants the lateral range of the two Pearson distributions are difficult to distinguish in 2D, an artificial case in Figure 2 is used to demonstrate.

Call for Questions

If you have hints, tips, solutions or questions to contribute, please contact our Applications and Support Department
Phone: (408) 567-1000 Fax: (408) 496-6080
e-mail: support@silvaco.com

Hints, Tips and Solutions Archive

Check out our Web Page to see more details of this example plus an archive of previous Hints, Tips, and Solutions
www.silvaco.com

Join the Winning Team!

Standardize your process/ device
and CAD design using Silvaco's

“TCAD Driven CAD™”

To get a demo and product description contact or visit a Silvaco office near you:

- Santa Clara
- Phoenix
- Austin
- Boston
- Guildford
- Grenoble
- Munich
- Tokyo
- Seoul
- Hsinchu

SILVACO

INTERNATIONAL

USA Headquarters:

Silvaco International

4701 Patrick Henry Drive, Bldg. 2
Santa Clara, CA 95054 USA

Phone: 408-567-1000

Fax: 408-496-6080

sales@silvaco.com

www.silvaco.com

Contacts:

Silvaco Japan

jpsales@silvaco.com

Silvaco Korea

krsales@silvaco.com

Silvaco Taiwan

twsales@silvaco.com

Silvaco Singapore

sgsales@silvaco.com

Silvaco UK

uksales@silvaco.com

Silvaco France

frsales@silvaco.com

Silvaco Germany

desales@silvaco.com

*Products Licensed through Silvaco or e*ECAD*

