# (IntelliSuite)

## IntelliMask, IntelliFab, AnisE

Intellisuite Training Manual, Intellisense

IntelliSuite

(IntelliMask)          (IntelliFab)          (AnisE)



# 1.        (IntelliMask)

## 1.1  Introduction

IntelliMask is specially designed for constructing and editing MEMS device level masks.

A. **<u>Starting IntelliMask</u>**

There are five ways to start IntelliMask: selectingConstruct…Layout…Create Mask from IntelliFab,selecting a Create/Edit menu item in the AnisE main user

interface, selecting the IntelliMask icon from the Windows Start menu, double clicking on the IntelliMask application, or double clicking on any mask file ("*.msk").

B. **Window layout**

The window layout is shown in Fig. 1. Many of the menu items are also represented in the toolbars,shown at right. The drawing area in the center of the window contains the 2D display of the mask.The area at the bottom of the window contains extra information to assist with editing the mask. At the top of this area is the layer number and the x- and y-coordinates of the mouse pointer in the drawing area. Thebottom line shows the current status of the application.
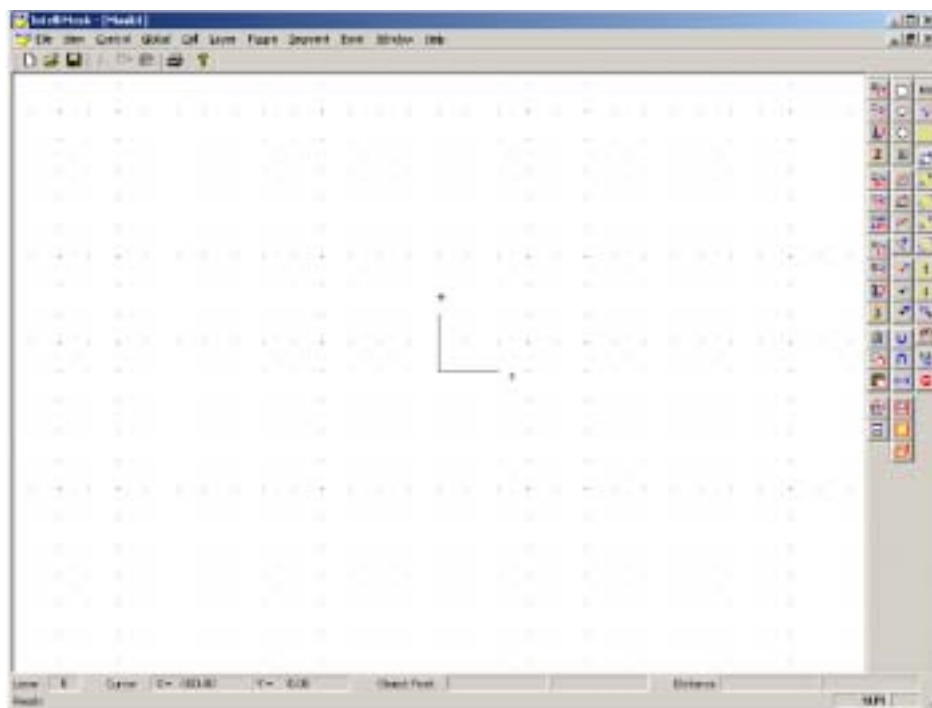


Fig. 1 Main window layout

## 1.2 Example

A. ***Comb-drive performance simulation***

(1). View area definition

First, we must define our view area.

**Click** *View…View Editor*

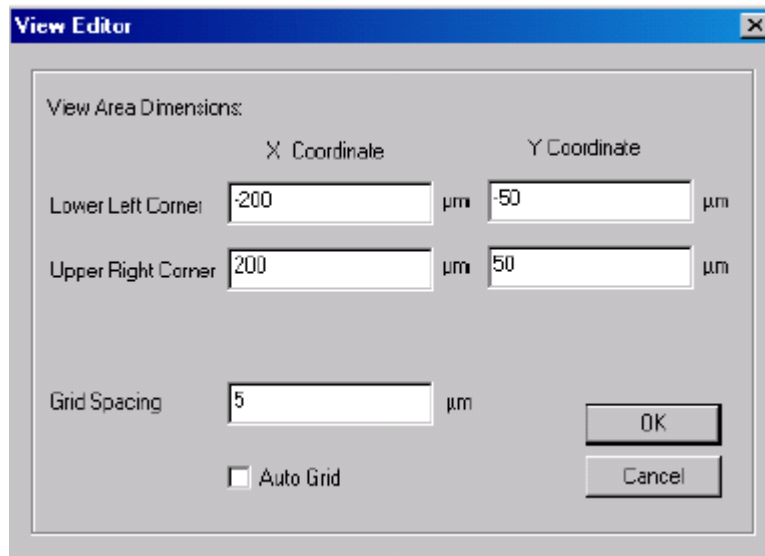Enter the values shown in the following figure.

Fig. 2   View Editor

## (2).   Figure creation

There are three phases to the creation of this mask: creation of the outer figures, creation of the center figures, and creation of the comb fingers.

## 1.   Outer figure creation

Start by creating the outer beam structures.

Click the Create Rectangle button:

Then, click the Keyboard button:

Enter one point at a time, clicking Apply (or pressing enter) after each. There are four rectangles, each defined by the two endpoints of a diagonal.

|  | Point 1 | | Point 2 | |
| --- | --- | --- | --- | --- |
|  | X | Y | X | Y |
| Rectangle 1 | -175 | 24 | -74 | 21 |
| Rectangle 2 | -175 | -1 | -59 | -4 |
| Rectangle 3 | -175 | -26 | -74 | -29 |
| Rectangle 4 | -175 | 24 | -170 | -29 |

Close the Keyboard dialog.

Next we will add the anchors.

Confirm that the Create Rectangle button is still selected: ▢

For these shapes, we will click and drag between the two points which define the rectangle's diagonal.Press the left mouse button at (-75, 35), drag to (-65, 10), and release. Then, drag from (-75, -15) to (-65, -40) to create the second rectangle.
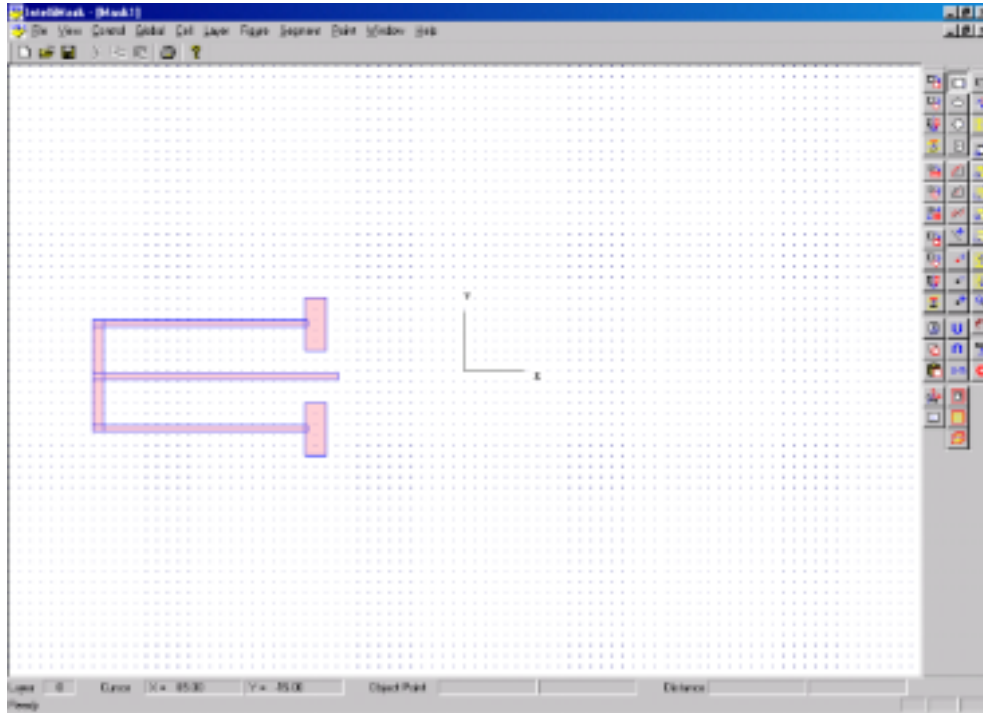


Fig. 3   Left side of comb drive

We will next copy these figures to create the right hand side.

Click the Select Figure button: ▢

Click and drag a rectangle around all six figures to select them.

Click the Copy Reflect button: ▢

Click and drag to create a line along the y-axis to define the line of symmetry/reflection. When the mouse button is released, the figures will copy to the right.

Click the Select Figure button: ▢in order to exit the Copy Reflect mode and enter the Select Object mode.

Deselect all of the previously selected figures by clicking and dragging a box around them.
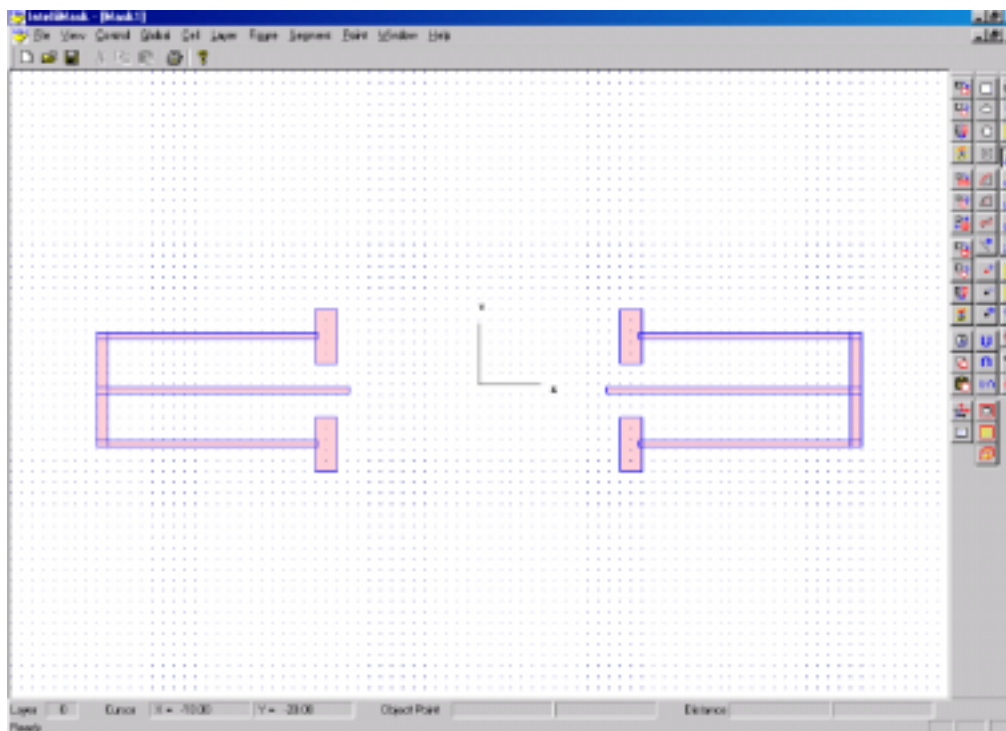
Fig. 4    Left and right sides of comb drive

## 2.    Center figure creation

To create the center figures, click the Create Rectangle button:

Click and drag from (-60, 5) to (60, -10). To create the second rectangle, click and drag from (-50, -35) to(50, -45).
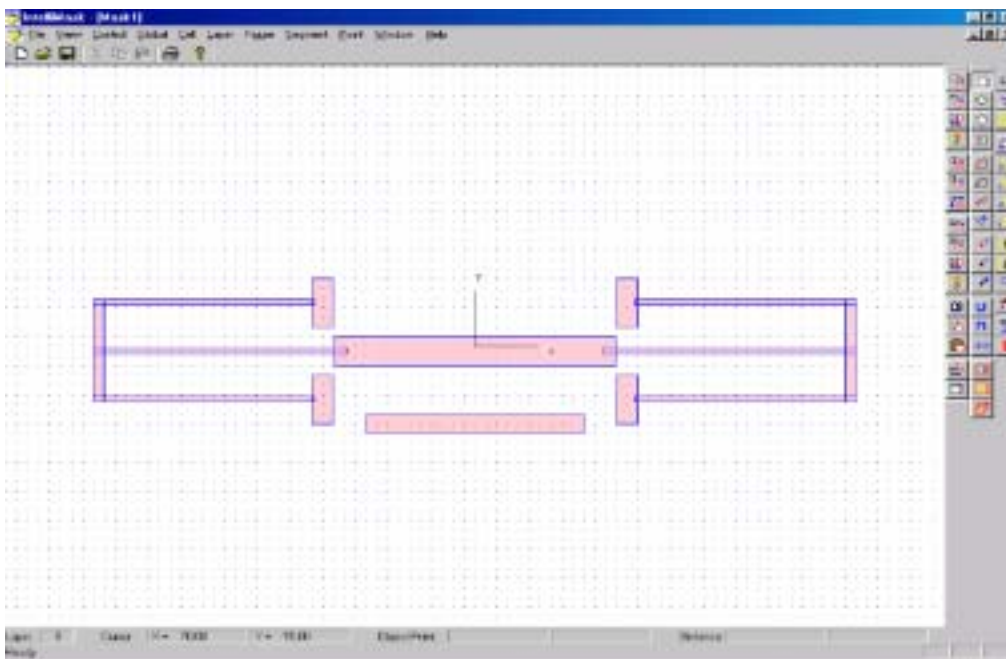
Fig. 5    Center rectangles

3.    Lower comb finger creation

We will create one comb finger first, then use the multi-copy feature to create the others.

Confirm that the Create Rectangle button is still selected:

Then, click the Keyboard button:

Enter (-38.5, -36) followed by (-36.5, -20). Close the Keyboard dialog.

Then click the Select Figure button:

Select the rectangle you just created by clicking inside the figure or by drawing a box completely around it.

Click on the M-Copy Translate button:

Click the Keyboard button:

The Multiple Translation Copy Definition window will appear. Enter 5 for the X separation and 15 for the Multiplicity. Click OK and the rectangles will be added.

Fig. 6    Multiple translational copy definition window

Click the Select Figure button:

Deselect the first small rectangle you created.

## 4.  Upper comb finger creation

Click the Create Rectangle button:

Then, click the Keyboard button:

Enter (-36, -25) followed by (-34, -9). Close the Keyboard dialog.

Then click the Select Figure button:

Select the rectangle you just created.

Click on the M-Copy Translate button:

Click the Keyboard button:

The Multiple Translation Copy Definition window will appear. Enter 5 for the X separation and 14 for the Multiplicity. Click OK and the rectangles will be added.
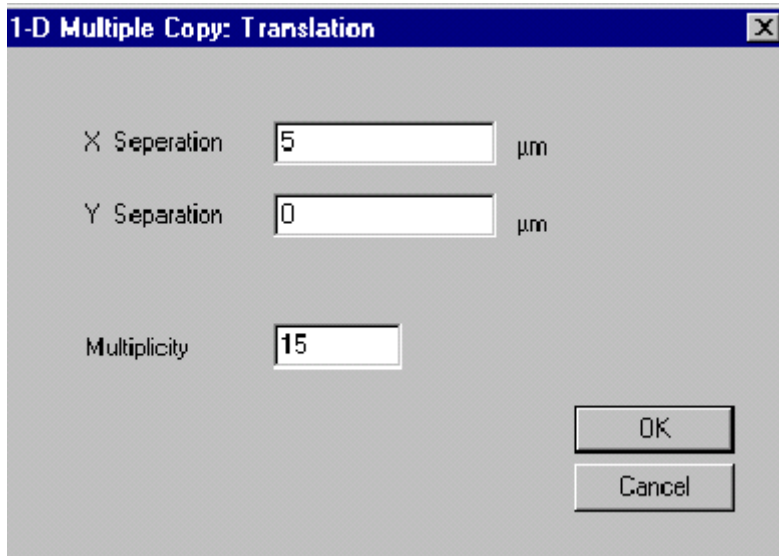
Click the Select Figure button:

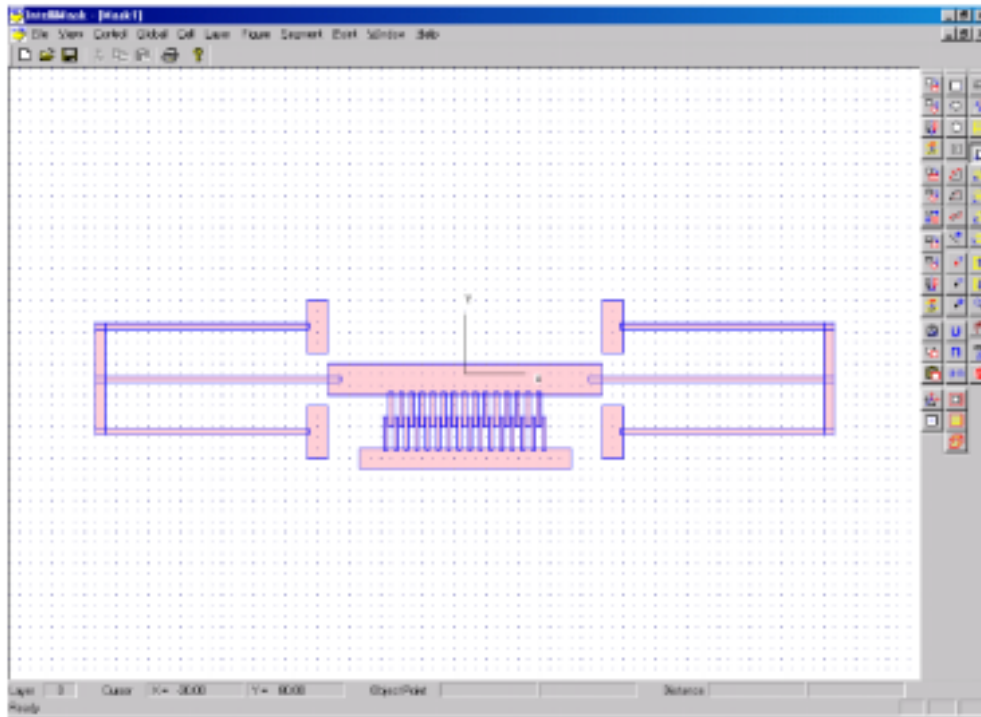Deselect the first small rectangle you created.

Fig. 7    Final mask

The comb fingers may appear to be misaligned and/or asymmetrical. If so, this is most likely an artifact ofscreen resolution. To zoom in, hold down the **Shift** key and press the **Down Arrow**. To zoom back out,hold down the **Shift** key and press the **Up Arrow**.

## (3).    Saving

Save the mask by selecting *File...Save*. In order to use a mask within IntelliFab, the file must be saved as *mask#.msk*, where # is the appropriate mask number. As this mask will be Mask 1, it must be saved as mask1.msk. Save the file in the `Training\comb-drive` directory.

## (4).    Footings creation

The footings are a subset of the current mask. Before we begin, we should save the current mask as Mask 2. Select *File...Save As...* and change the filename to mask2.msk. This will prevent us from accidentally overwriting Mask 1 later.

Click the Select Figure button:

Select everything except the five rectangles shown in the next figure, either by clicking inside each figure or by drawing a box completely around them.
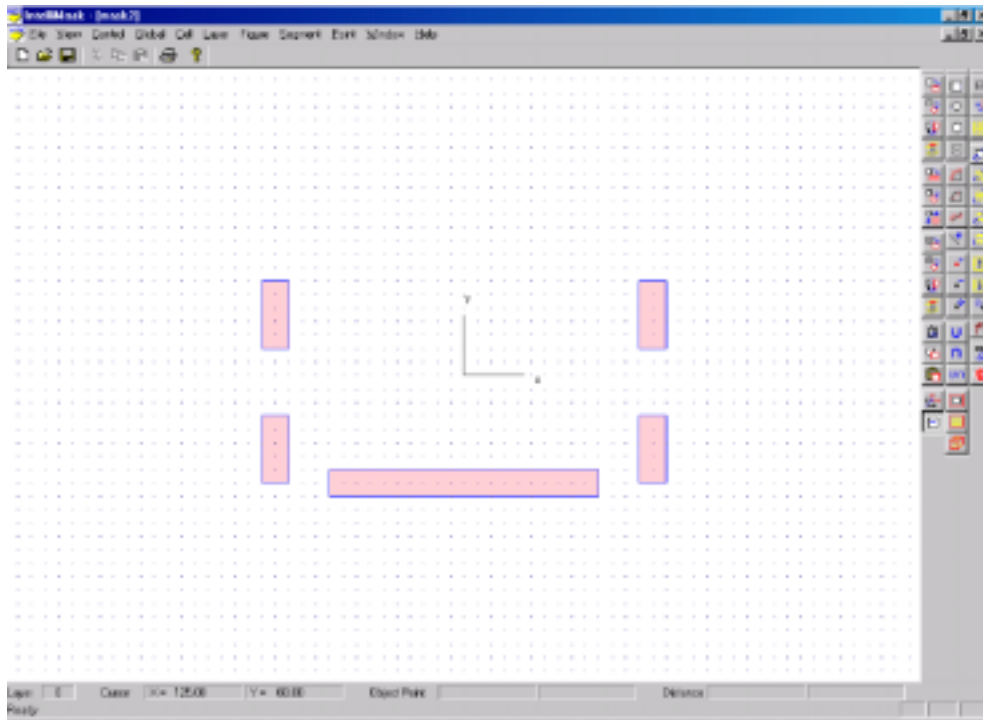
Then click the Delete button:

Fig. 8   Footings mask

Save the mask by selecting *File…Save*.

## (5).   Exiting

Exit the mask editor by selecting *File…Exit*.


## B.   ***Generation of complex masks using cell hierarchy***

When developing a new MEMS design, generating the mask layout for a single device is only part of the challenge. The actual masks that will be used in fabrication must cover an entire wafer, often encompassing dozens (or hundreds, depending on wafer and device sizes) of device-level layouts on a single mask. Once the wafer-level mask has been laid out, it can be difficult to effect minor changes in device-level masks, due to the sheer number of changes that must be made on the entire wafer.

This example will describe the procedure for generating a complex mask using cell hierarchy, in which changes can be easily propagated from feature-level to device-level to wafer-level.

We will begin by creating a mask consisting of a single comb drive, then create an array of cells to create a mask for 200 comb drives, and finally, create a wafer-level mask.

## (1).   Starting IntelliMask

To begin, open the IntelliMask software program.

Click Start…Programs…IntelliSuite…IntelliMask

## (2).   Generation of comb-drive cell

The mask for a single comb-drive has already been generated. It is the same mask that was used in the Comb-Drive Performance Simulation example in Section 2. We must first identify this mask as a cell.

Click *Cell…Open…*

Select the file `comb.msk` from the `Training\mask-cells` folder. The comb-drive mask should appear, centered at the origin.

### 1.   Creation of comb-drive matrix

We will create a grid of comb drive devices, in which each individual comb drive will be an exact copy of the comb drive at the origin. Subsequent changes in the original comb drive will be reflected in all of the copies.

Click Cell…Copies…2-D Array

A dialog box will appear. Enter 400 for the X Separation, 200 for the Y Separation, 10 for the number of Columns, and 20 for the number of Rows. Click *OK*. You can allow IntelliMask to auto-rescale the grid.
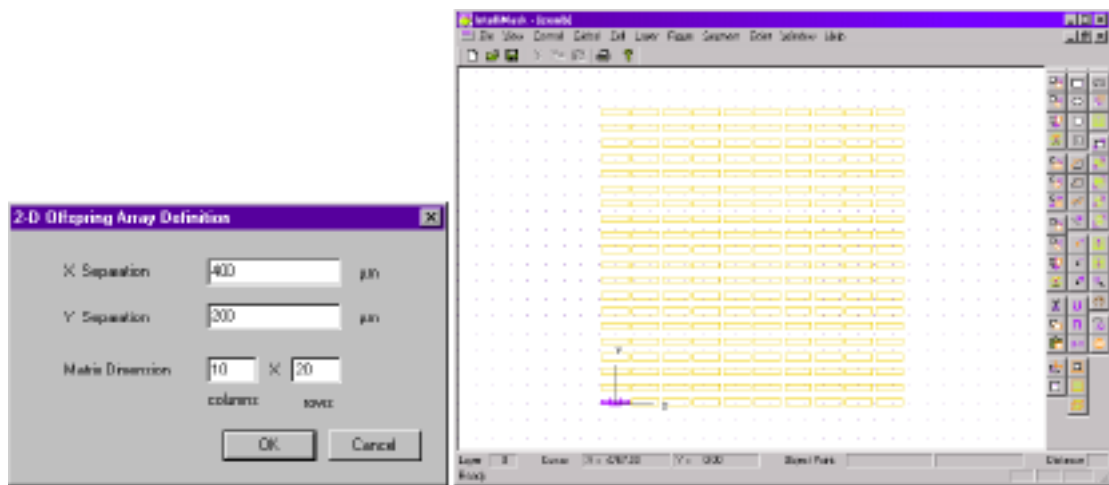


Fig. 9   2-D Array dialog box and resulting array of footprints

The IntelliMask window will automatically zoom out, and will show rectangles enclosing the footprints of all of the comb drives in the array. We have now finished generating the array.

Click *File…Exit and Save*

## (3).   Creation of wafer mask

We can now begin creating the mask for the entire wafer. We will start by inserting a grid of comb drives centered at the origin. An occurrence of a cell within a mask is referred to as an *instance*.

### 1.   Insertion of cell instance

Click Cell…Instance

Select the `comb.msk` file. IntelliMask will prompt you for the location of the cell's origin in the new mask. For example, if we specify that the cell origin should be at the origin of the wafer mask, the bottom left comb drive would be located right on the origin. We want the center of the array on the origin.

Click the Keyboard button:

Enter (-1800, -1900). The grid of comb drives should appear. Close the Keyboard dialog box.

Click File…Save As…

Save the mask as `wafer.msk`.

This entire array is still only occupying a 4mm x 4mm area of the wafer. We could, in reality, insert several more instances of the array in the wafer level mask. For demonstration purposes, however, we will use only the one instance.

### 2.   Creation of wafer-level mask features

In addition to the individual device masks, there may be additional features on the wafer-level mask. These features may include:

- Alignment features
- Logos
- Test structures

We will add a few basic features. First, we must resize the drawing area.

Click View…View Editor Enter (-3500, -3000) for the Lower Left corner, (3500,3000) for the upper Right corner, and 200 for the Grid Spacing.

Click the Create Circle button:

Click and drag from (-3200, 1400) to (-2600, 1400). Create a second circle by clicking and dragging from (-3200, -1400) to (-2600, -1400).

Click the Create Rectangle button: Click and drag from (2400, 1600) to (3000, -1600).
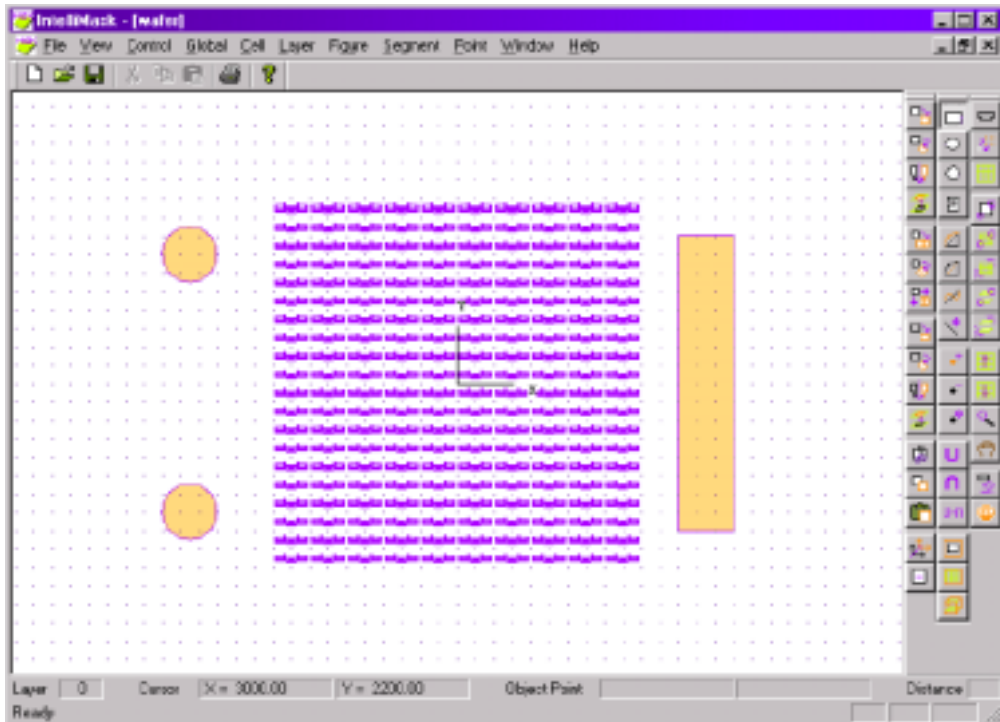
Fig. 10   Wafer-level mask

Although it is not necessary, it is a good idea to save at this point.


(4).   Modification of comb drive mask

We want to change the length, width, and number of comb fingers on the comb drive device. In our relatively simple wafer-level mask, there are 200 comb drives, each with 29 comb fingers. Obviously, modifying 5800 comb fingers would be an extremely tedious and difficult task. Fortunately, we will only need to modify the fingers on one comb drive.

First, we must re-open the comb-drive cell mask.

**Click** *Cell…Open…*

Select the comb.msk file. We will zoom in around the "master," or original comb drive.

**Click** *View…View Editor*

Enter (-200, -50) for the Lower Left corner, (200, 50) for the Upper Right Corner, and 5 for the Grid Spacing.

Click the Select Objects button:

Click and drag a box around all of the comb fingers. Click the Delete button:

We will now draw a new set of comb-fingers.

Click the Create Rectangle button, and click the Keyboard button.

Enter (-44.5, -36) and (-39.5, -15). The first comb finger should appear (see Fig. 11a). Close the keyboard dialog box.

Click the Select Objects button and select the comb finger.

Click the M-Copy Translate button:

Click the Keyboard button. Specify an X Separation of 12 and a Multiplicity of 7. The bottom combfingers are now complete (see Fig. 11b).

Click the Copy Translate button:

Click the Keyboard button. Specify an X Translation of 6 and a Y Translation of 6 (see Fig. 11c).

Click the Select objects button, deselect the first comb finger, and select the new upper comb finger. Click the M-Copy Translate button and the Keyboard button. Specify an X Separation of 12 and a Multiplicity of 6. The comb drive has now been updated (see Fig. 11d).
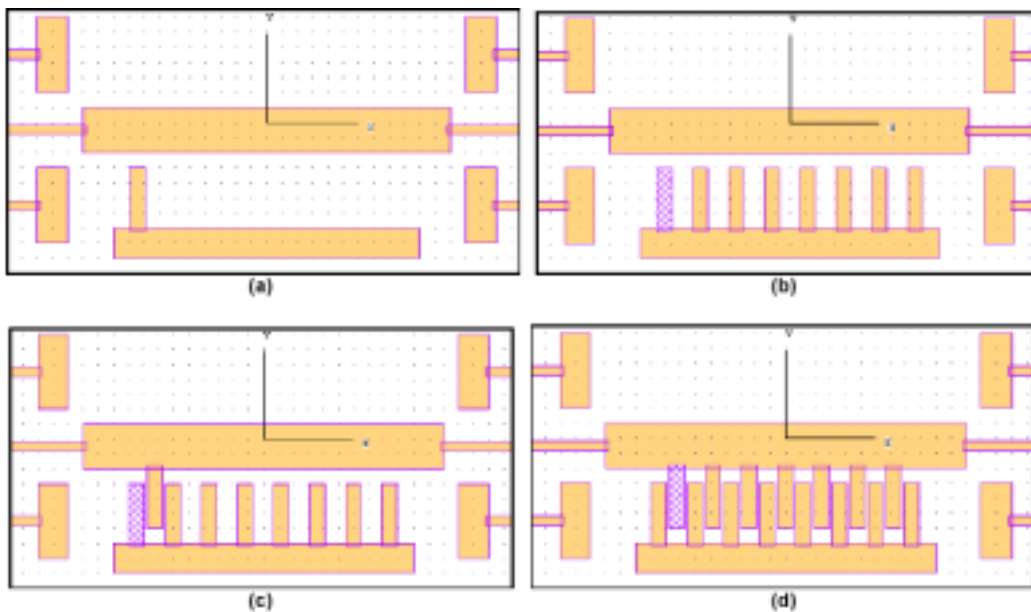


Fig. 11    Creation of new comb fingers

**Click** File…Exit and Save

This will save changes to the "master" layout in comb.msk, as well as entire arrayed comb.msk instance.

The wafer-level mask (wafer.msk), reflecting the changes that were made, will now be displayed.

(5).    Inspection of wafer-level mask

Because of the small size of the individual comb drives, you most likely will not be able to discern the change in the comb fingers.

Click the Zoom button: 

Click and drag a box around a few of the comb-drives. You will see that the comb fingers have been updated in all of the comb drives.
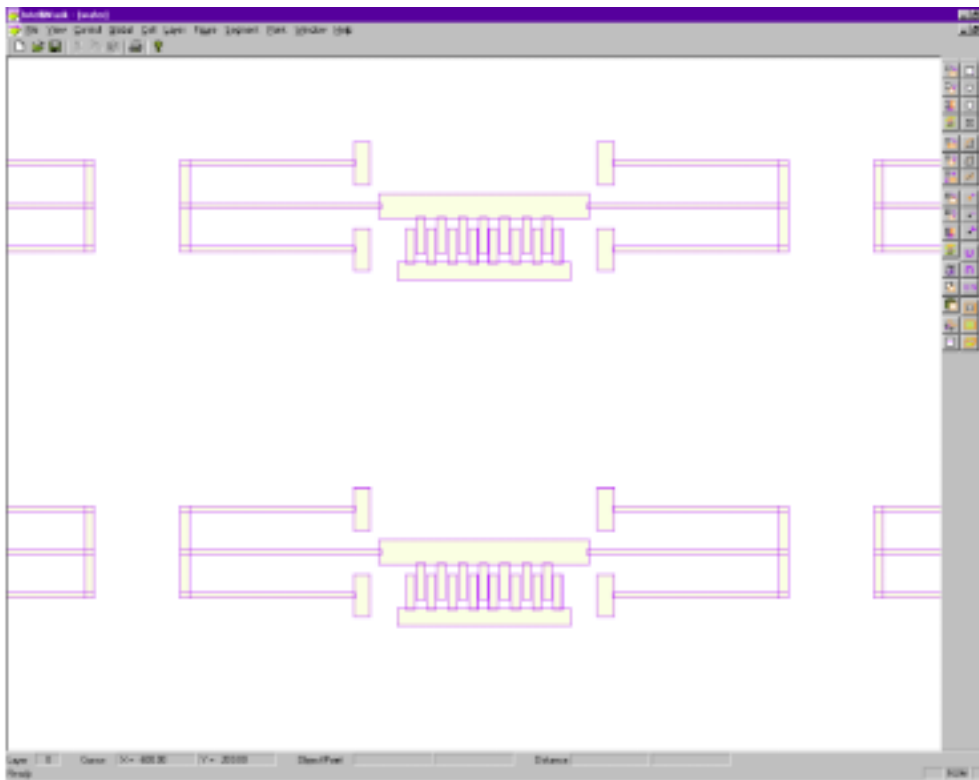


Fig. 12   Zoomed comb drives showing updated comb-fingers

# 2.          (IntelliFab)

## 2.1  Introduction

IntelliFab allows you to construct 3D models directly from the process steps and then export them to the analysis modules.

To open IntelliFab, select the following from the main Windows menu bar.

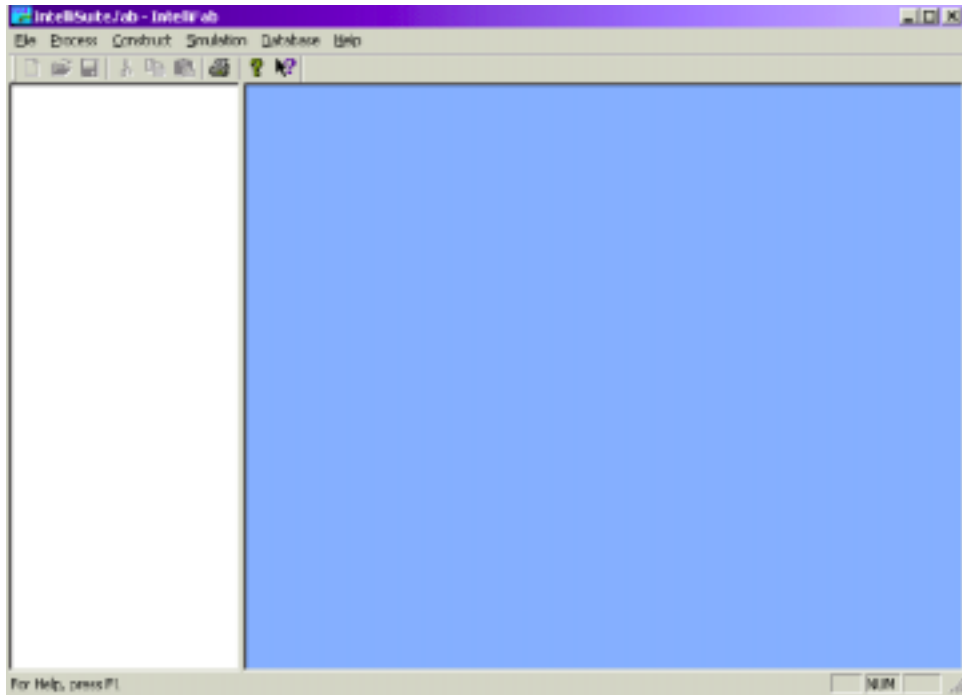**Click** *Start…Programs…IntelliSuite…IntelliFab*

Fig. 13    IntelliFab main window

## 2.2  The IntelliFab window

There are three main areas in the IntelliFab window: the pulldown menus, the selection area, and the process table.

### A.  **Pulldown menus**

This is the horizontal menu situated at the top of the grid area. It displays submenus that activate various process flow options.

### B.  **Selection area**

The selection area is on the left of the window. This area displays all available process steps.

Note: To widen the selection area (and make the process table narrower) move the pointer over the vertical grey dividing line and click and drag horizontally.

### C.  **Process table**

The process table is constructed in the large window on the right of the screen. In this area, you will build your process flow from steps chosen from the selection area.

© 2004

## 2.3  Example

A.  **Build and visualize the MEMS structure based on process**

A MEMS 3D solid model can be built in IntelliSuite by defining a list of the process steps used for manufacturing. This process sequence is referred to as a Process Table. IntelliSuite provides Process Table templates to users, or they make their own by accessing the IntelliSuite database of individual process steps.

(1).  Building the process table

**Click** Start…Programs…IntelliSuite…IntelliFab
Click in the Database Area, on the white, left of the window.

**Click** File…Open Database
Select the matfab file, and four items will appear.

Position the pointer over the gray dividing line between the selection area and process table. Click and drag until the selection area is about two inches (five centimeters) wide, for better visualization.

**Click** *Definition…Si…Czochralski*
Then, double click on *100* to open the *Process Dialog* window. Note the wafer specifications as shown in the upper left of the Process Dialog window that appears. Click *Apply Changes* to add this step to the process table.

**Click** *Etch…Si…Clean*
Double click on *Piranha*. Click *Apply Changes* to add this step to the process table. Now we can start depositing material on the silicon substrate.

**Click** *Deposition…SiO2…Thermal*
Double click on *Wet*. Click *Apply Changes* to add this step to the process table.

**Click** *Deposition…Si3N4…PECVD*
Double click on *Ar*. First, we will change the film thickness (t_film) to 400 nm. To do this, double click on *t_film*. This will bring up a dialog box in which you can enter the new value. Click *Apply Changes* to add this step to the process table.
Now, let's take a look at the structure that we are building. In the *Process Area* (blue section), click on the Nitride process step (the last one).

**Click** Construct…Visualize
You will be asked to save the file. Go to Training\comb-drive. Enter start_comb for the filename, and click the Save button.The structure is visualized through the highlighted process step, by default.

To rotate the structure, click and drag the mouse in the window area on the left, or use the keyboard arrow keys.

Other visualization options are also available in the 3D window. In the Visualization window, click theright mouse button to change to either Translate or Zoom Window. Zoom Window is a box zoom. Click and drag until the area you wish to see appears in the box. To zoom out, hold down the control button

while clicking and dragging the mouse. You can also zoom in and out using the keyboard by holding down the Shift key and pressing the down and up arrows, respectively. Click Cancel to exit the 3D View. At this point, we will import a saved IntelliSuite process template that contains the entire process sequence

for the comb drive device. Click once in the process table area (blue).

**Click** File…Open Fab

Change to the IntelliSuite directory and then select the Training directory. Open combdrive\perfect_comb file. The complete process table will appear.

You will see two "Definition UV Contact Suss" process steps. These reference the masks previously created and precede etching steps. Double click on either of the "Definition UV Contact Suss" to open the process step. Then click *Layout* to see the mask in IntelliMask. Exit from the mask editor. Click *Cancel* to close the Process Dialog box.

**Click** *Construct…Visualize*

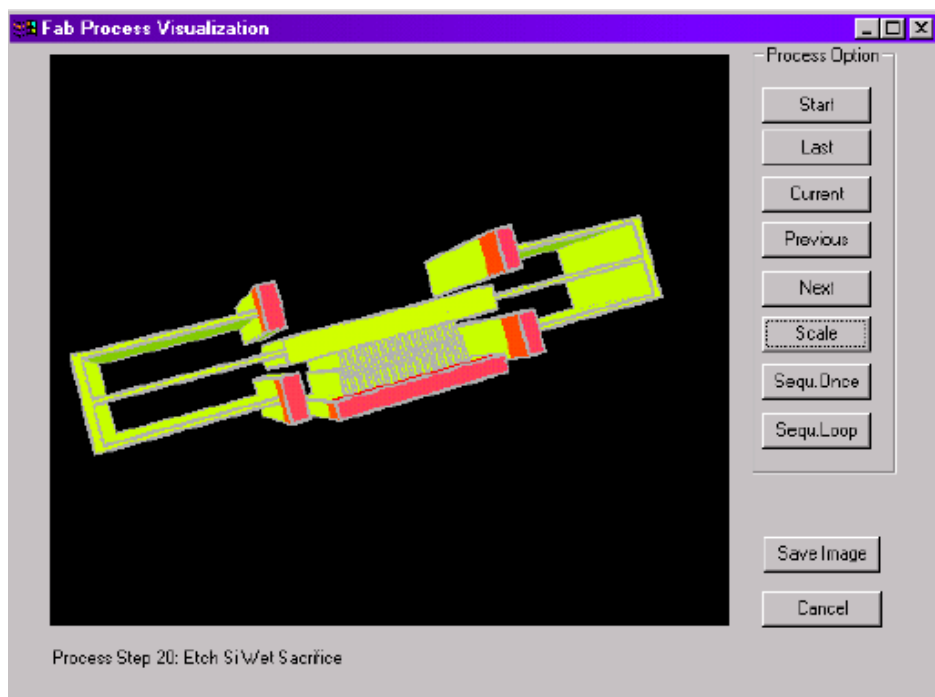Click *Last* button to display the figure below.



Fig. 14   Comb structure

Click Scale and input 2 for the z-dimension scale factor for better visualization.
Click *Cancel* to exit the 3D View.

## (2). MEMaterial: determining thin-film material properties

You can see any of the process data by double clicking on that step in the process table, for example steps three or four.

Once the process for building the structure is complete, it will be helpful to study the details of the process steps that we have created. A very important set of parameters in MEMS simulation is the thin-film material properties. IntelliSuite and its MEMaterial analysis module allow the user to characterize the thin film material properties based on actual machine settings during fabrication. Accurate material properties are necessary for accurate simulation results. Errors in material properties are directly proportional to errors in final simulation results.

We will investigate MEMaterial's capabilities with respect to the silicon nitride deposition step in our process table.

Double click on step four: Deposition Si3N4 PECVD Ar.

The process dialog will appear. Once again, the machine settings for this process step can be seen in the upper-left portion of the window. To investigate the relationship between these machine settings and material properties, click on the *Material Property* button. MEMaterial will appear.

**Click** *View…2D Graph*

Then click on the property button:

**Click** *STRESS*

Click *OK*. Next click on the parameter button:

**Click** *T_dep*
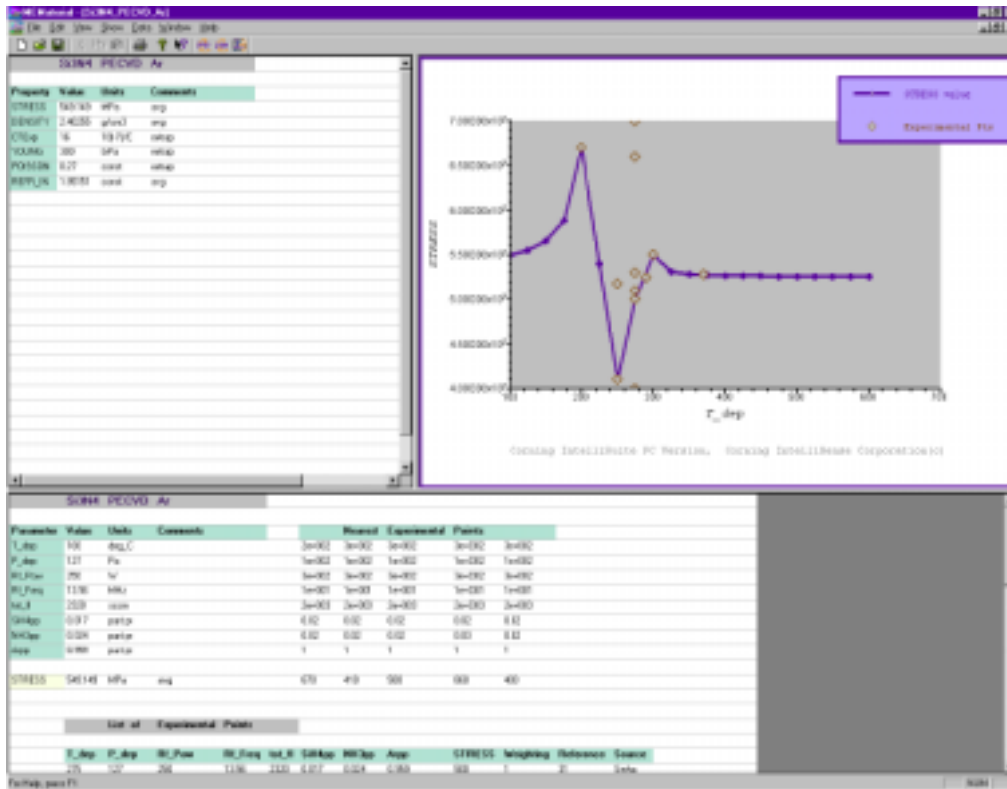
Click *OK*. The following results will appear:

Fig. 15    2D MEMaterial stress results

The blue line represents the 2D plot of the particular property and parameter, while the red dots represent variation in the property due to variation in other parameters.

Note: You can view the data for this particular deposition step if you use the scroll bar (at the lower right) and scroll down.

In order to view a 3D representation of the a material property plotted versus two machine parameter:

Click *View…3D Graph*

Then click on the property button:

Click *STRESS.*

Click OK. Next click on the parameter button:

Click *T_dep*

Hold down the control button and click *P_dep*. Click OK.
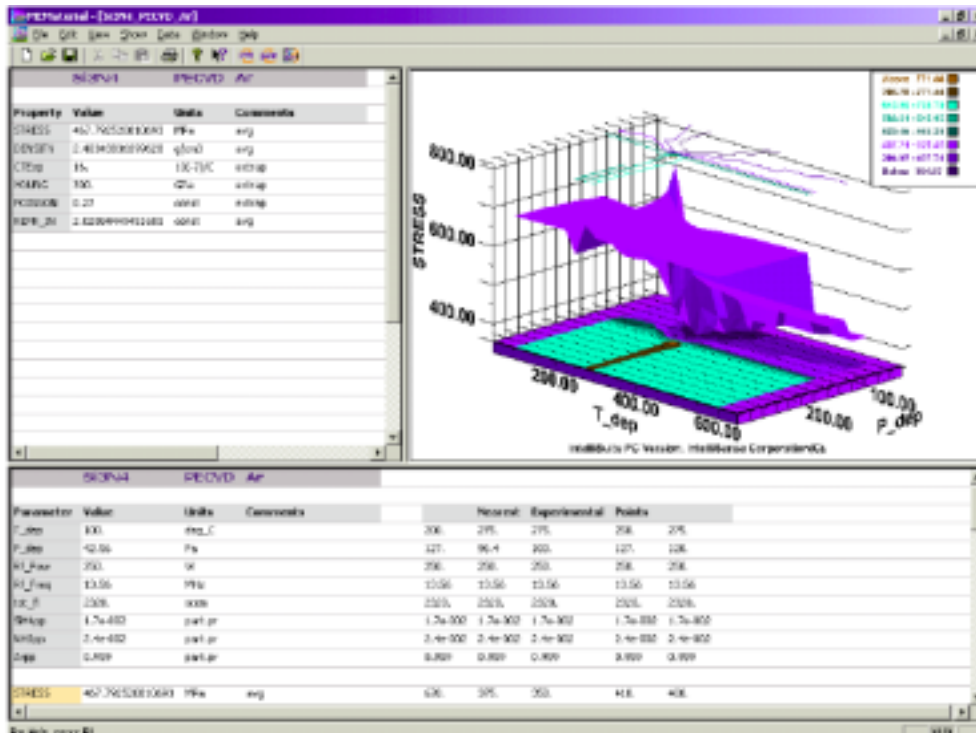
The following results will appear:

Fig. 16    3D MEMaterial stress results

The 3D graph can be rotated by clicking and dragging in the graph window.

**Click** File…Exit

This will return you to IntelliFab. Click Cancel in the Process Dialog box to return to the main IntelliFab window.

# 3.            (AnisE)

## 3.1  Introduction

This section will take you through different examples. Each example builds on the knowledge gained in the previous example.

There are four examples:

Example 1a: Anisotropic one sided etching of <100> Wafer

Example 1b: Etching with negative mask; <110> Wafer

Example 2: Anisotropic two sided etch

Example 3: Etching with misaligned masks – mask manipulation

Enter AnisE. The main AnisE window will appear. When you start the program, the filename will default to AnisE1. This will be changed when the file is saved. Click *File…Save As* and save the file as `Training\AnisE-examples\Example1a`.
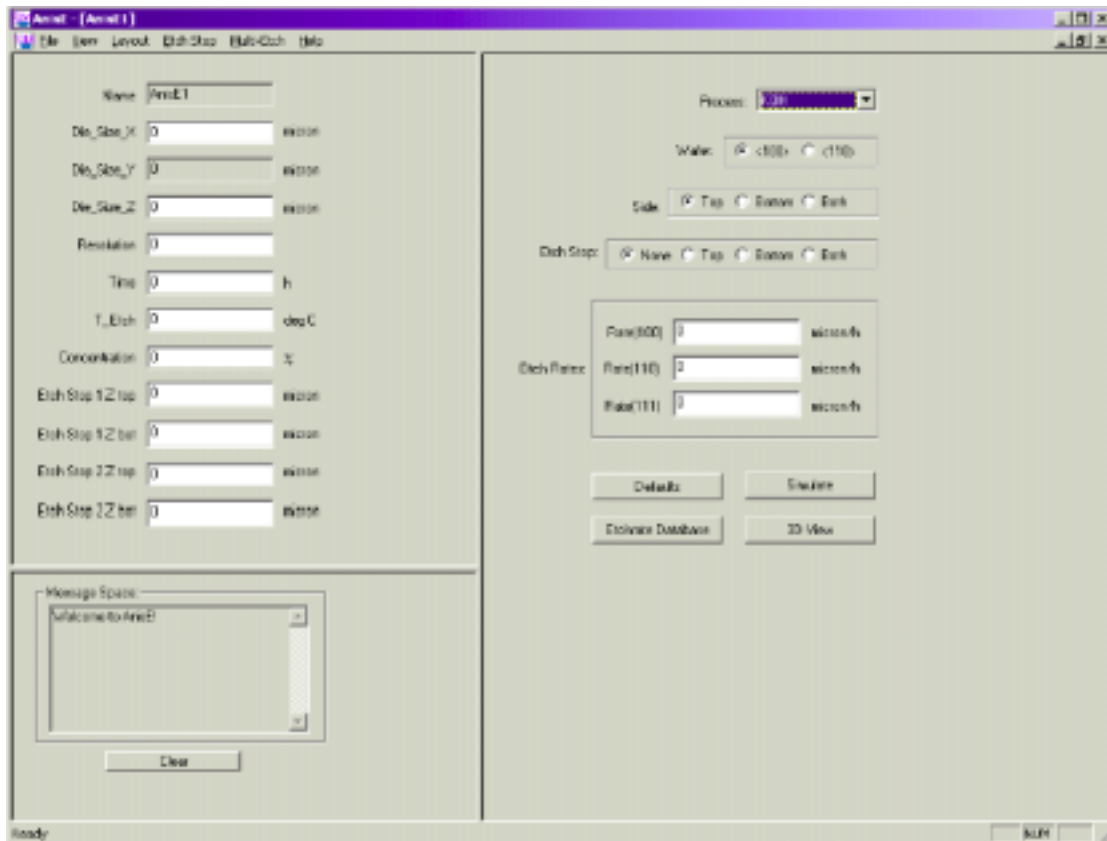
Fig. 17    AnisE window

## 3.2  Example

### A.  **Example 1a: Anisotropic one sided etching of <100> wafer**

In this example we will:

э create a single mask layout

э define etching parameters

э simulate the anisotropic etching process

э visualize the results

First we will create the mask layout. For this example, we will use the Mask Editor.

### (1).   Mask editor

Move the pointer to the *Layout* menu.

**Click** *Layout…Create/Edit…Top Mask*

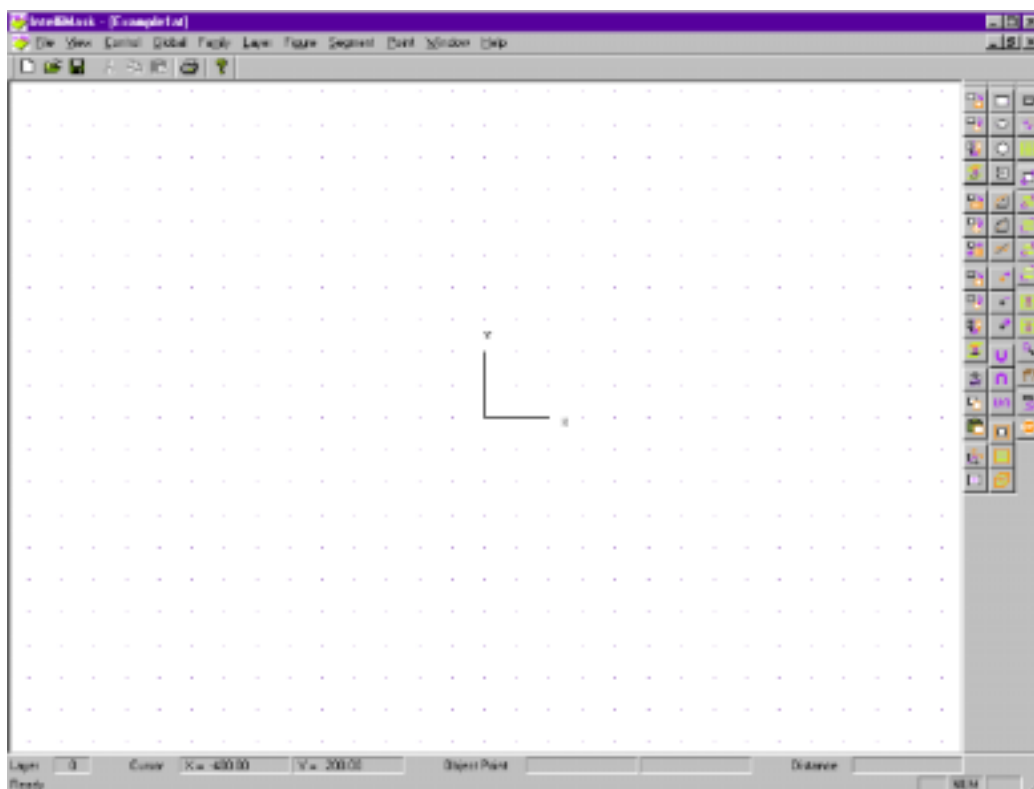The Mask Editor will appear.

Fig. 18    Mask Editor window

1.    Grid spacing

**Click** View…View Editor

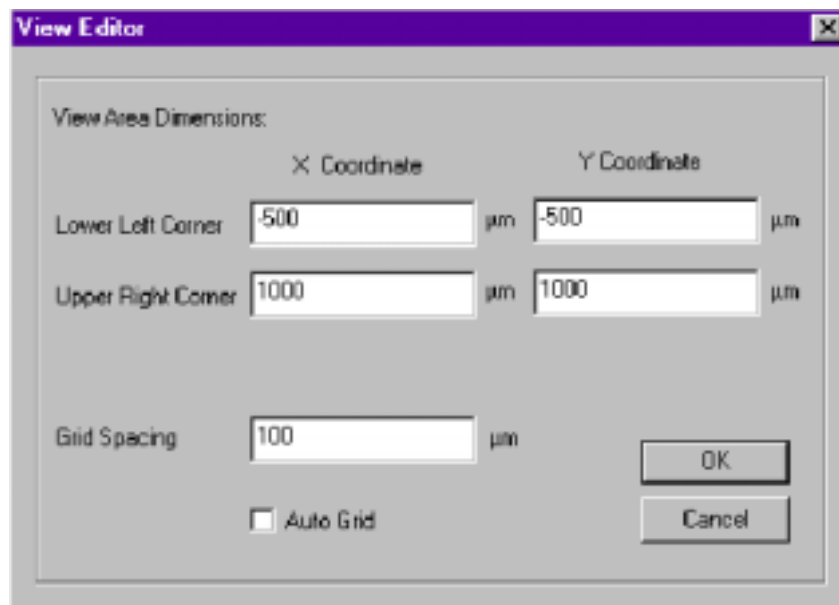A View Area Specification box will appear (Figure ).



Fig. 19    View Editor box

Enter 100 in the Grid Spacing box. Change the Upper Right Corner to have the coordinates 1000,1000.

Then Click *OK*.

2.  Creating rectangles

To create a rectangle in the mask, click on the Rectangle toolbar button:

Click and drag to create the following four rectangles:

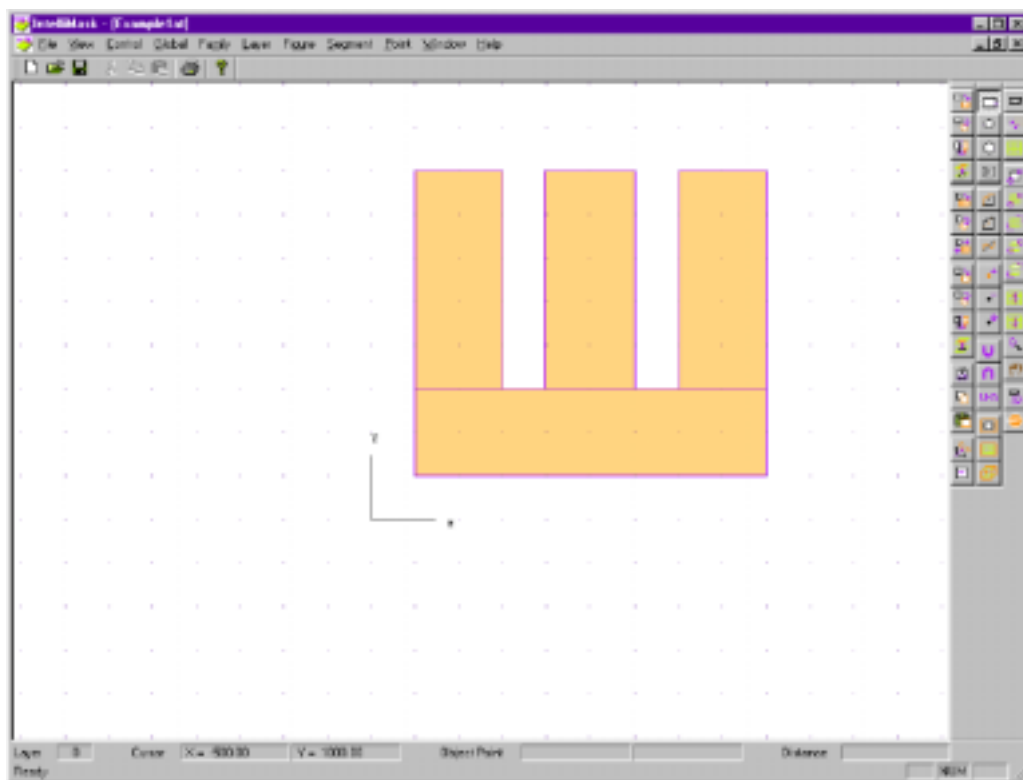| | X | Y |
|---|---|---|
| Set 1 | 100 | 300 |
| | 300 | 800 |
| Set 2 | 400 | 300 |
| | 600 | 800 |
| Set 3 | 700 | 300 |
| | 900 | 800 |
| Set 4 | 100 | 100 |
| | 900 | 300 |



Fig. 20    Final mask

**Click** File…Save

IntelliSuite will automatically assign a filename of Example1at.msk.

© 2004

**Click** File…Exit

## (2). Parameters box

Now enter values into the Parameters Box:

- Die_Size_X 1000
- Die_Size_Z 300
- Resolution 200
- Time 4
- T-etch 70
- Concentration 30

In the upper right corner of the AnisE window, choose KOH from the Process box list.Once all of the values have been entered, click the *Etchrate Database* command button in the lower right corner of the window.

Note: The program defaults to the <100> wafer orientation, etching of the top side of the wafer,and no etch stops.

## (3). Simulation

Now that the data has been entered, we are ready to simulate.

Click the *Simulate* command button. During the simulation, a window will appear with a 2D view of the etched structure as the etching proceeds. When the simulation is complete, this 2D window will disappear automatically.

Once we have run the simulation, we can use the *3D View* button. Click the *3D View* command button. A 3D representation of our structure will appear showing the effects of our etch process (Figure ). The new window is entitled "3D View."
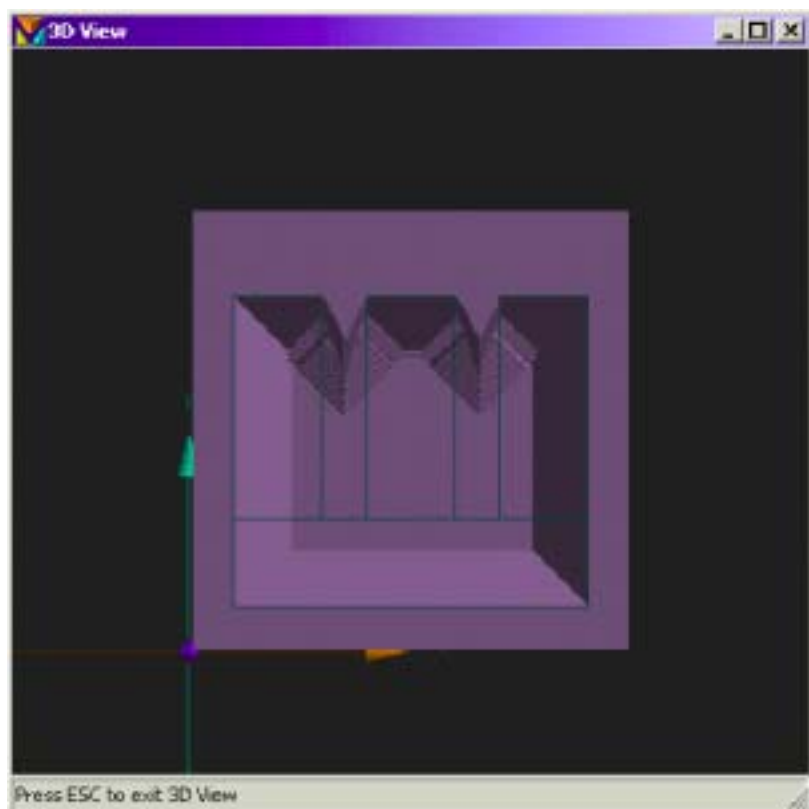
Fig. 21    Final etched image

The etched die can be manipulated, rotated, translated, and zoomed using the mouse buttons. You may change any of the parameters in AnisE and run the simulation again to see the effects of the change on the final etch result.

(4).   Saving results

To save the file, click *File…Save*.

B.   **Example 1b: Etching with negative mask; <110> wafer**

In this example we will:

- import a mask and specify etching outside the mask
- define <110> wafer and other etching parameters
- simulate and visualize the anisotropic etching process
- save and load simulation settings

(1).   Import a mask

   **Click** File…Save As

Change the filename to Example1b.

   **Click** Layout…Import…Top Mask

Select the previous mask (Example1at.msk) for import into AnisE.

Next, we will specify that etching is to proceed outside the mask (i.e. specifying the negative of the mask).

**Click** Layout…Mask Option…Etch Outside

Select wafer <110> from the right portion of the AnisE window.

Enter parameters as follows:

- Die_Size_X 1400
- Die_Size_Y 1400
- Die_Size_Z 300
- Resolution 200
- Time 2.5
- T-etch 70
- Concentration 30

**Click** *Simulate*

The simulation will show the etching process in 2D over the 2.5 hour etching period in increments of 27minutes. This window disappears when the simulation is complete.

**Click** *3D View*

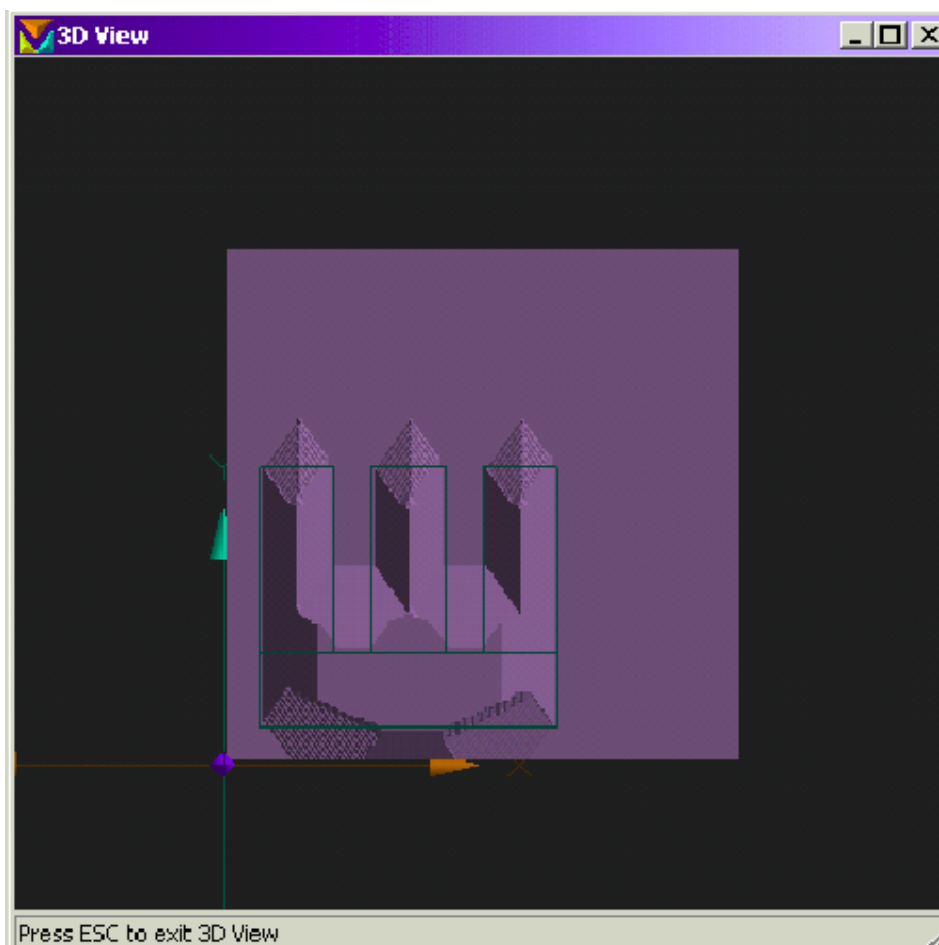The 3D View window appears, containing the 3D results.



Fig. 22    3D view of outside etching

Close the 3D View window by pressing **esc**.

(2).  Saving the results

> **Click** File…Save
> This will save the file in the entry in the Name field.

## C.  **Example 2: Anisotropic two sided etch**

In this example, we will:

> э read in saved data and then modify etching parameters
> э import one mask and create a second mask
> э simulate and visualize the anisotropic etching process
> **Click** *File…Save As*

This will allow you to rename the file. Change the name to Example2. The filename will change in the main AnisE window.

Change the following parameters:

Left:

> Concentration 20

Right:

> Wafer <100>
> Side Both

Update the etch rates by,

> **Click** *Etchrate Database*

The etch rates are now updated, and the AnisE window will look as follows.
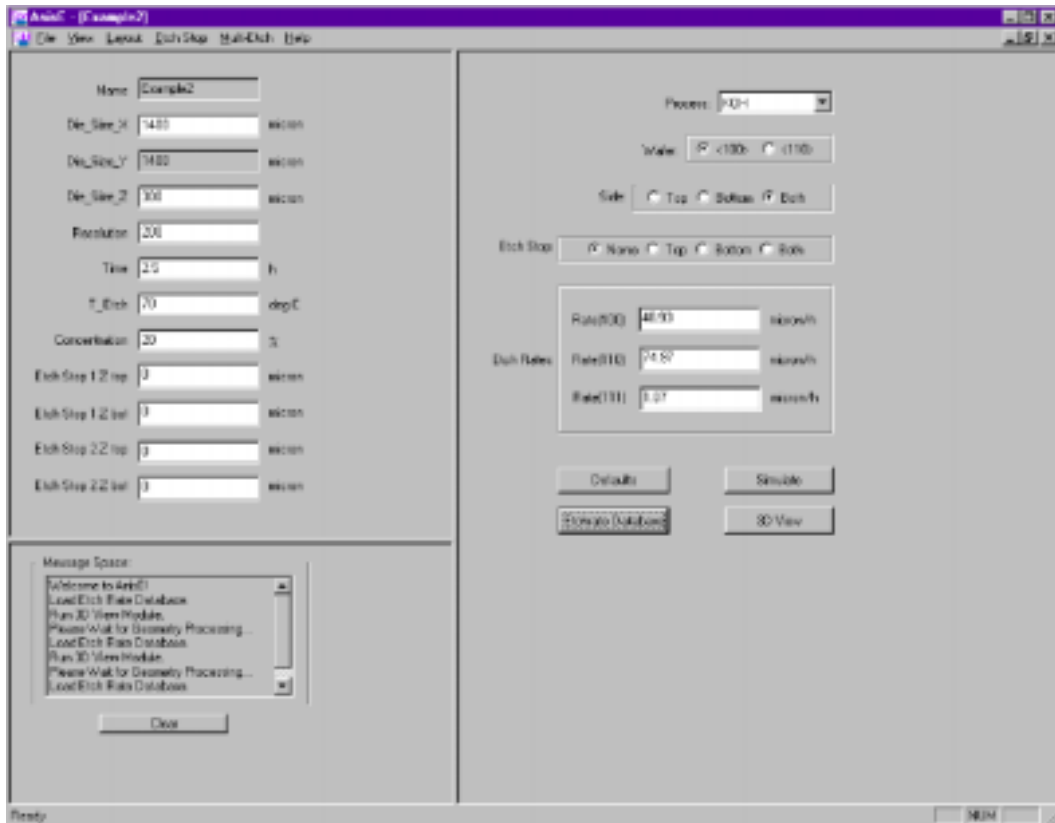
Fig. 23    Updated AnisE window

## (1).   Import a mask

The mask created in Example 1a will be imported as the top mask.

**Click** *Layout…Import…Top Mask*

Select the Example1at.msk, then click *Open*.


## (2).   Creating another mask

We will now create the bottom mask.

**Click** *Layout…Create/Edit…Bottom Mask*

The Mask Editor will open.

## (3).   Mask editor


## 1.   Set grid size


**Click** View…View Editor

In the Window Setup box that appears, enter 50 in the Grid Spacing box. Also, change
the Lower Left Corner to (0,0) and the Upper Right Corner to (1200,1200).

Click OK

## 2. Edit new mask

Select the Rectangle toolbar button. Click and drag to create a rectangle with the following coordinates:
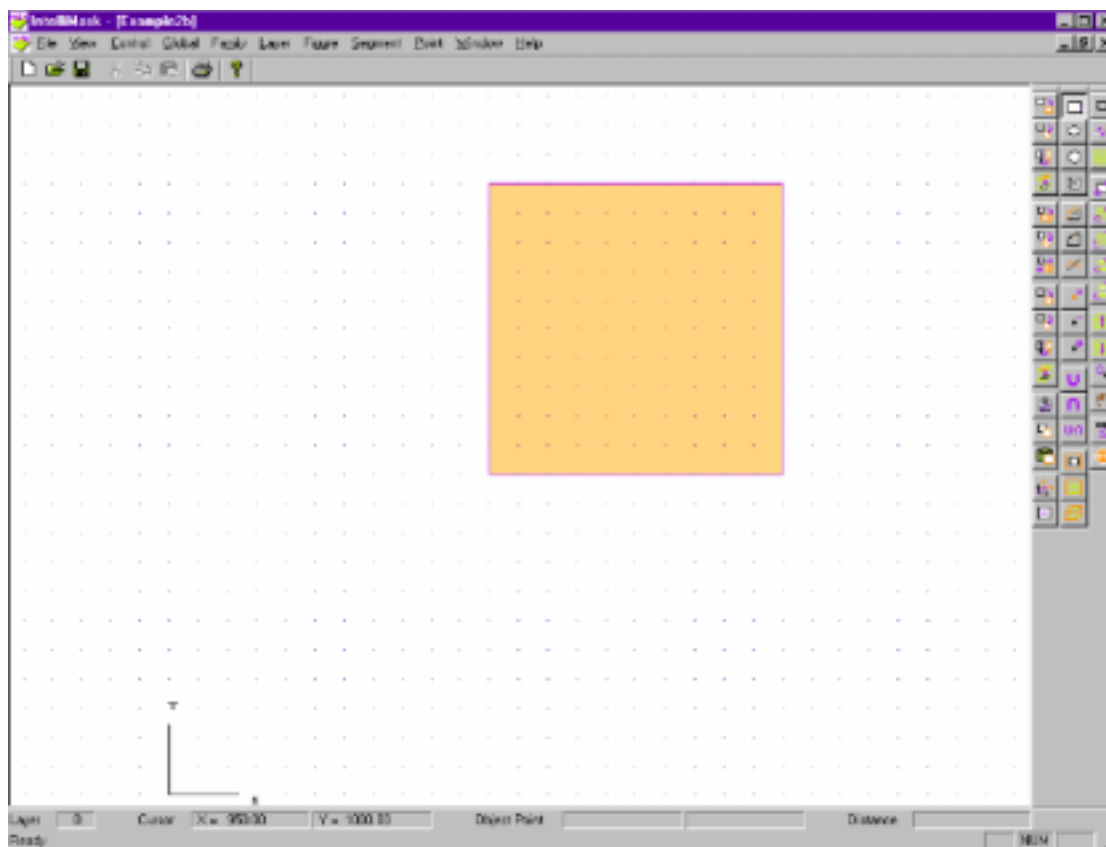
Corner 1 (550,550)

Corner 2 (1050, 1050)



Fig. 24 Mask Editor window

**Click** *File…Save*

**Click** *File…Exit*

This will return you to the main AnisE window.

## (4). Etching

Specify the etching to be performed inside the mask.

**Click** *Layout…Mask Option…Etch Inside*

## (5). Simulation

**Click** Simulate

You will only see the top mask during the simulation.
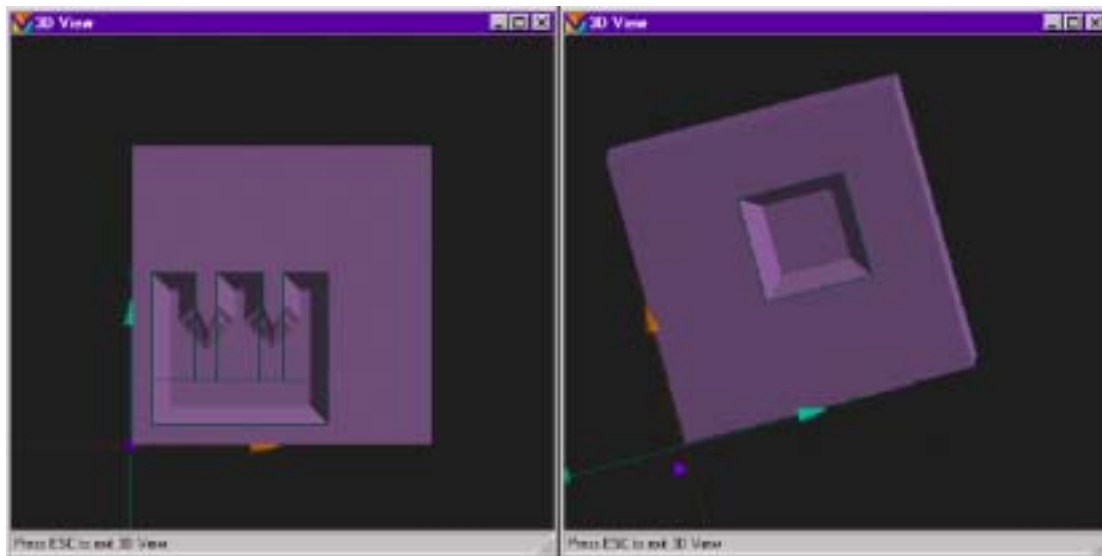
(6).   Visualization

**Click** 3D View



Fig. 25   3D View results

Close the 3D View window by pressing **esc**.

(7).   Multietching

Now we will continue etching this structure until "breakthrough" occurs.

**Click** *Multi-Etch…Load Previous*

Now we can change etching conditions and/or etching parameters. Change the etch time to 1.0 hour.

**Click** Simulate

The 2D window disappears when the simulation is complete.
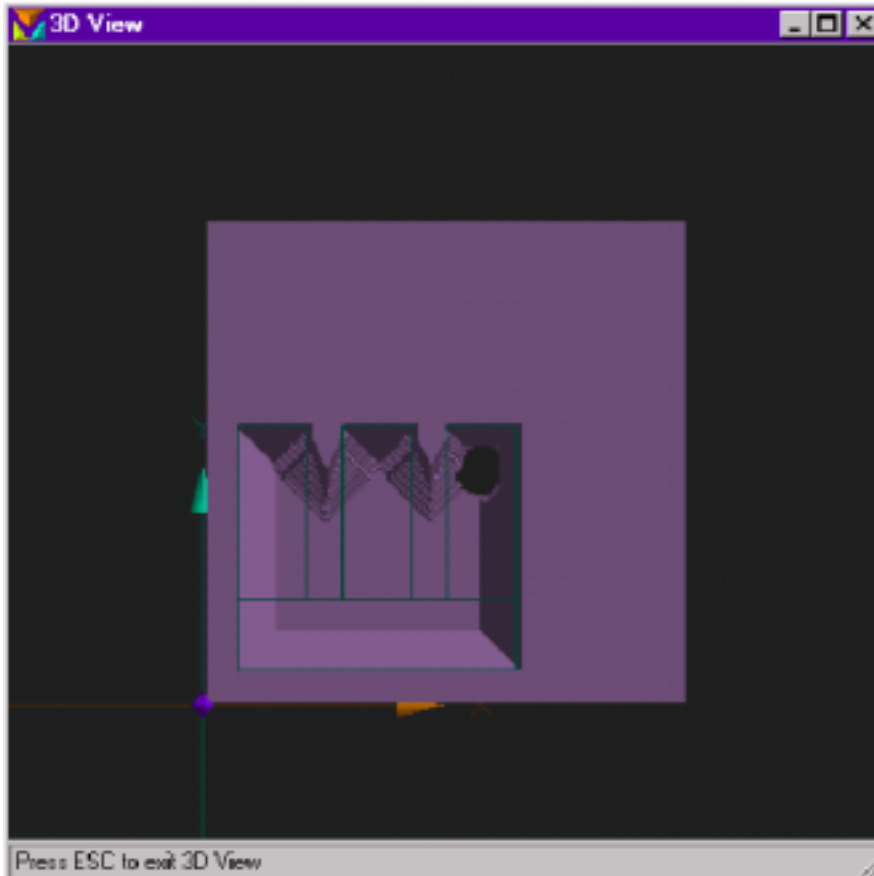
**Click** 3D View

Fig. 26    3D View of multi-etch

D. **Example 3: Etching with misaligned masks and mask manipulation**

AnisE can be used to simulate the effect of mask misalignments on your final etch structure. These results can be used to determine process tolerance.

In this example, we will:

- read in saved data and modify etching parameters
- import the top mask
- manipulate the mask by rotating
- simulate and visualize the anisotropic etching process

(1).   Setting the etching parameters

We will work from the Example 1b settings, but modify them as follows.

**Click** File…Save As

Enter `Example3` when prompted.

**Click** Multi_etch…No Previous

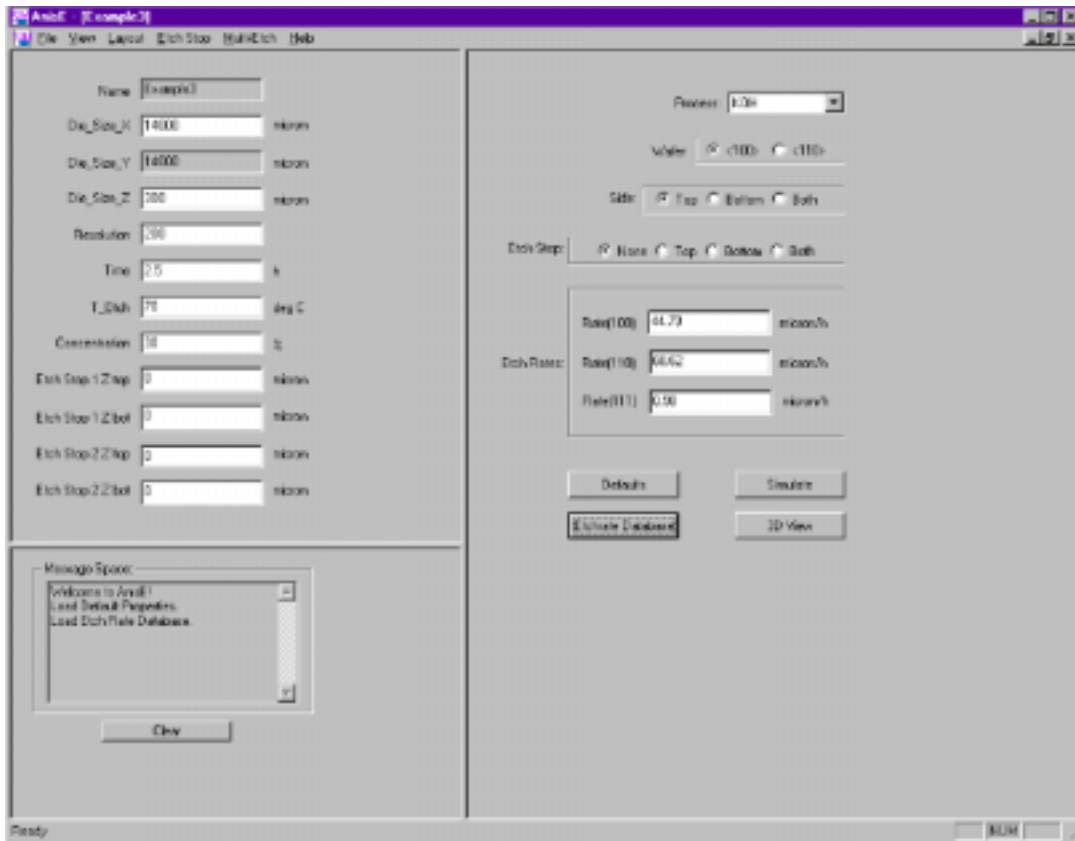Change the time back to 2.5 hours., and change the "Side" setting to Top.

Fig. 27   Updated AnisE window

(2).   Import the top mask

**Click** Layout…Import…Top Mask
Select the Example 1 mask.

(3).   Manipulate the mask

Mask manipulation can be used to determine process tolerances. We will rotate and stretch the top mask in the x-direction.

**Click** *Layout…Create/Edit…Top Mask*
The Mask Editor will open, showing the current top mask.

Click on the Select Layer button:

Then click on the Move Rotate button:

Finally, click on the Keyboard button:

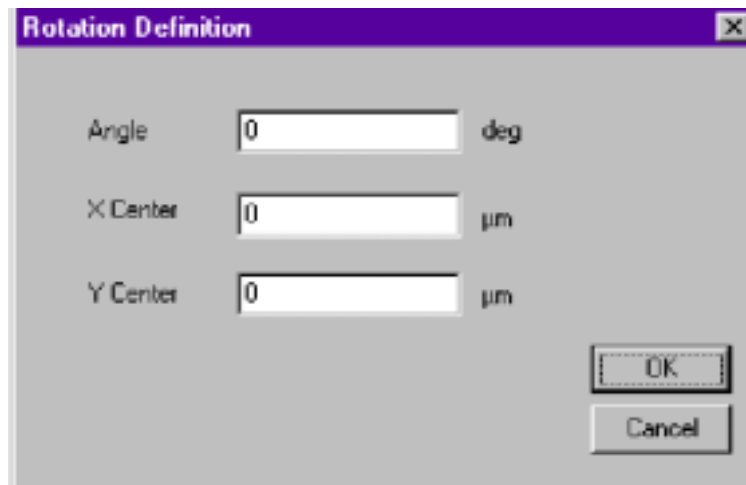The Rotation Definition Dialog will appear. Enter 10 for the Angle.

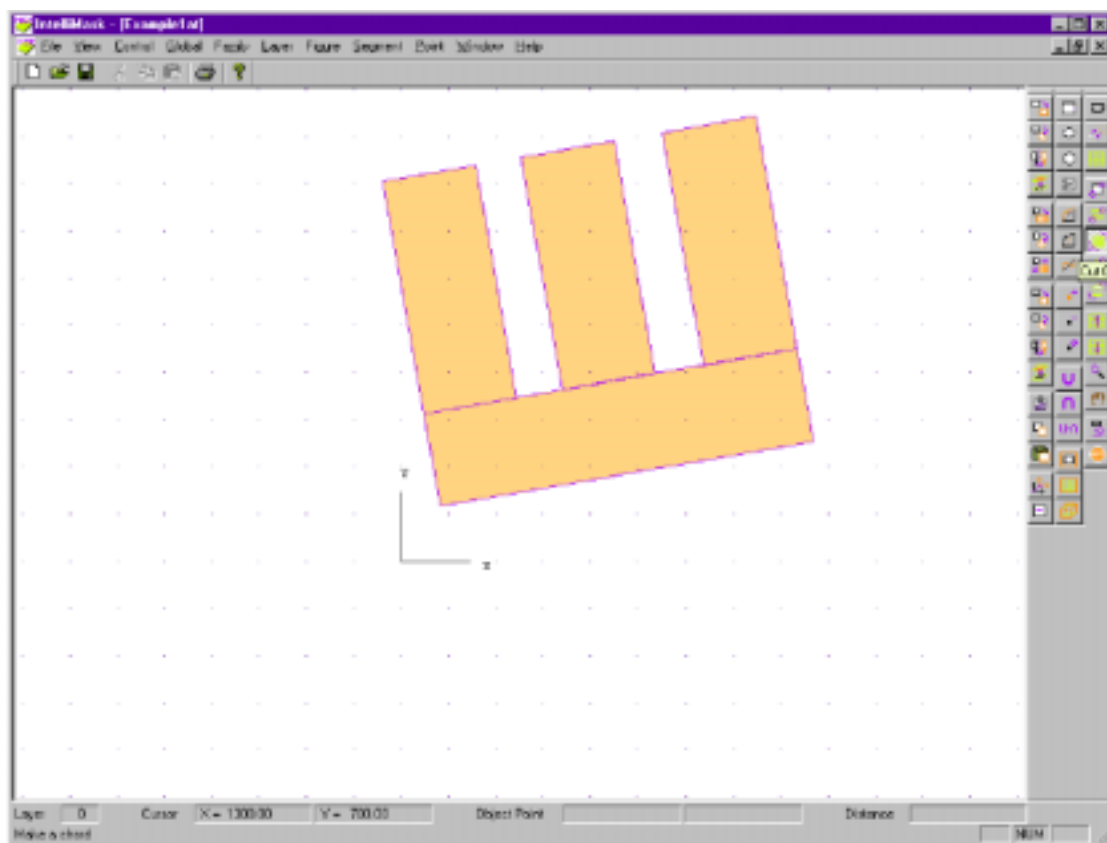Fig. 28   Rotation definition dialog box



Fig. 29   Rotated mask

**Click** File…Save

**Click** File…Exit

The mask manipulation feature allows for easy and quick mask handling. It is always possible to create the mask within AnisE (as shown above) or by importing it in

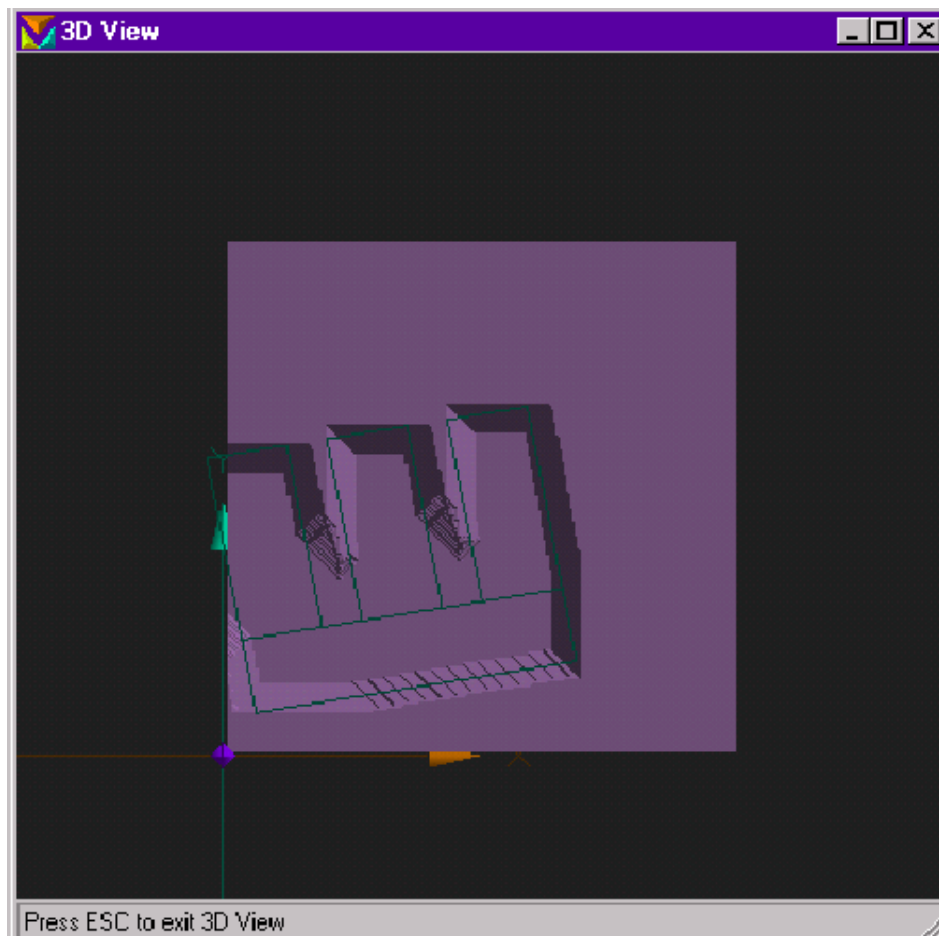GDSII or DXF formats through the Mask Editor.Specify etching to be performed inside.

**Click** Layout…Mask Option…Etch Inside

(4). Simulation

**Click** Simulate

(5). Visualization

**Click** 3D View



The above figure shows the effect of mask misalignment. Compare this result with the result where no rotation and no mask stretching was assumed; it will show significant differences.

Press **esc** to leave the 3D View.

**Click** *File…Exit*