ECSE-4250 IC Processes and Design

Fall 2007

TMA TSUPREM4 INSTRUCTIONS

You can run tsuprem4 from any terminal on campus. Following is a brief summary of instructions. If you have any problems, please email bhati@rpi.edu or walsht2@rpi.edu

1. How to run TMA tsuprem4 from any RCS workstations?

    a)   Login to a workstation supported by RCS or any other departments.

    b)   Login to your account (your RCS account) and open a unix window.

    c)   In window, type **xhost +**

    c)   Then type **ssh -l (your_login_id) rce-01.ecse.rpi.edu**. If it asks a question, type "yes". Then enter your password.

    d)   Type **tsuprem4 filename** where the filename is the file you created. The file should be a text file.

 2. How to print files?

        To make postscript plot files, add the following line to the plot statement in your tsuprem4 input file.
        Option   plot.out=filename.plt device=postscript where filename.plt is any output filename. You can print this postscript file using any postscript printer. If you do not use this statement, the graph appears in your screen.

3. The best way to learn TSUPREM4 is by running some example files. Go to the example directory by typing the following command:
   cd /cad/synopsys/Z_2007.03/taurus-tsuprem4_2007.03.0/examples/tsuprem4

   To make it easier, we created a directory called "examples" in your home directory. You can go to this directory and look for example files. Copy any of these files to your home directory and run them.

   Use the command **ls** to list the files in the directory. You can run any files with the *.inp* extension. To run the files you will have to copy them to your home directory (*/home/yourusername*) using the **cp** command. For example if you want to copy the file *s4ex1a.inp* from the example directory to your home directory then type:
   cp s4ex1a.inp ~   (assuming that you are in the examples directory)

   On unix/linux systems, the tilde (~) is shorthand for home directory. This command will copy *s4ex1a.inp* from the current directory to your home directory. You will also need to copy the *TS4.config* file to run properly:
   cp TS4.config ~
   Use **cd** command to move around different directory. For example, to move to examples

directory from your home directory, type **cd examples**

To run the file, return to your home directory by typing:
cd ~

And run the file:
tsuprem4 s4ex1a.inp

If you wanted to organize your home directory, you can create directories using the **mkdir** command, for example:
mkdir examples

will make a directory called "examples". You can then move your example files into that directory:
mv s4ex1a.inp examples/
mv TS4.config examples/

Note that examples directory has already been created for you.
To get to that directory from your home directory you would use **cd**:
cd examples/

and to get back to your home directory:
cd ..

TSUPREM -4 TUTORIAL

Chapter 1: Introduction to using TSUPREM-4
Chapter 2: Statements and their Syntax
Chapter 3: Sample codes

ECSE-4250: IC PROCESSES AND DESIGN
Rensselaer Polytechnic Institute

CHAPTER 1

1.1 INTRODUCTION

TSUPREM-4 is a computer program for simulating the processing steps used in the manufacture of silicon integrated circuits and discrete devices. TSUPREM-4 simulates the incorporation and redistribution of impurities in a two-dimensional device cross-section perpendicular to the surface of the silicon wafer.

1.2 Getting Started with TSUPREM4

TSUPREM 4 is loaded on the **ecse-cad** machines. You can access this machine from anywhere. You must have the **user id** and **password** set up before you can login. The command to run the program is tsuprem4

1.3 ENTERING THE INPUT FILE

To enter the code for the input file, you have to use a text editor like *vi* or *emacs*. Appendix gives you a brief summary of working with the same. (Alternatively, you can create the file in your own PC and transfer it to the workstation)
After entering the code, save the file as *<filename.inp>*, to specify it as input file.

1.4 RUNNING THE PROGRAM

To run the program, just type

tsuprem4 <filename.inp >

The statements contained in the input file are executed and the output is plotted on the screen. TSUPREM 4 stores the output as **filename.out** in the home directory. The plots can be saved using **XV or Snapshots** utility.

1.5 Capturing the output image with XV or Snapshot utility:

The plots obtained on executing the input files can be saved using two methods

Using XV utility :

1. Type *'xv &'* on your terminal console to start the application.
2.   Right click on the 'xv program window' that opens up and select **grab.**
3.   Put in *5 sec* delay, hit **autograb**, and move to your display window that contains the
      plot.
4. The image will be grabbed in the specified time and the program will beep to indicate when
      it's done.
5. You can alternately choose grab, left click to get the window or draw a box with the middle
      mouse button to select a rectangle you wish to view. You should see a copy of the captured

image on the screen.

6. Save the file as *<filename.gif>* and then you can later 'ftp' it to your RCS account to print it
    out.

Using Snapshots utility

1. **Right click** on the blank screen, select **Applications   Snapshot**.
2.   Select the **Hide window During Capture** button and then left click on **Snap**.
3. Move the cursor to the **'AvanWaves'** plotting window and left click.
4. Wait for several seconds until the control panel reappears with an 'Image Viewer' window of
    the captured plot screen.
5. Select **File   Save As** in the Image Viewer window, pick a **'File Format'** (TIFF is
    best for PC and Macintosh files), enter a filename and select **OK**. This method will
    work for capturing an image of any window on the Sun workstation.

1.6 PRINTING THE OUTPUT

The input, output and plot files are now in your **ecse home directory**. To take a printout, you
have to **ftp** them to your RCS account and print it out from any printer connected to RCS
network.

1.7 PROCEDURE TO FTP FILES

At the command prompt in your machine, type

ftp ftp.rpi.edu

You will be prompted to enter your RCS USER ID and RCS PASSWORD
After verification, you will get a prompt as follows

ftp>

1. To transfer files from your current directory to RCS machines, just type
        ftp> *put < filename.inp>*
   To transfer multiple files say, all input files,
        ftp> *mput <*.inp>*
   The program will indicate that transfer is complete.

2. To transfer file from remote host to local machine,
        ftp> *get <filename.inp>*
   To transfer multiple files of same file extension,
        ftp> *mget <*.inp>*
    The program will indicate completion of transfer.

3. To quit from ftp, just type

ftp> *quit*

Your original command prompt will be restored

4. To copy a file from the machine to floppy,

    mcopy <filename.inp> a:

5. To copy from floppy to the machine,

    mcopy a: <filename.inp>

6.   To list files in floppy drive,

    mdir a:

—

-

CHAPTER 2

2.1 SOME USEFUL SIMULATION STATEMENTS

The most commonly used statements in TSUPREM 4 are:

1. INITIALIZE
2. LINE
3. SAVEEFILE
4. DEPOSITION
5. ETCH
6. IMPLANT
7. DIFFUSION
8. EPITAXY
9. SELECT
10. PRINT.1D
11. PLOT.1D
12. PLOT.2D
13. LABEL

The purpose of each statement, its syntax and the most commonly used parameters of each of the above statements are described below.

2.2 Syntax of parameter lists:

1. <n> and <c> represents parameter type to be numerical, character respectively.
2. Optional parameters are specified in 'square brackets []'.
3. When one of a group of parameters is to be chosen, the choices are separated by 'vertical bar |'.
4. Comment statements are preceded by '$' character and are used to document the input file.
5. The inputs can be in lower case or upper case or a mixture of the two.

TSUPREM 4 models the continuous physical processes using finite element and finite difference techniques .The initial structure is defined as a rectangular region of arbitrary or user-defined width and depth and is divided into mesh of overlapping triangular elements. Solution is calculated at the mesh nodes at the corner of these triangular elements.

2.3.1. INITIALIZE

The INITIALIZE statement sets up the mesh from either a rectangular specification or from a previously saved structure file. This statement also initializes the background doping concentrations of the impurities specified.

INITIALIZE
{ (IN.FILE=<c>   [SCALE=<n>]) | [WIDTH=<n> [DX=<n>]] [{<111> | <110> | <100> |

ORIENTAT=<n>}]
}

[IMPURITY=<c> { I.CONC=<n> | I.RESIST=<n>}]
[MATERIAL=<c>] [ANTIMONY=<n>] [ARSENIC=<n>] [BORON=<n>] [
PHOSPHOR=<n>]

IN.FILE   Name of saved structure file, scaled by SCALE. If omitted, a rectangular
                 mesh is generated.
WIDTH   Width of the initial structure and grid spacing DX in x-direction.
ORIENTAT   Crystalline orientation of the silicon substrate.
IMPURITY   Material of initial structure, default silicon. Also, the concentrations of
                 other materials can be specified.

2.3.2 LINE

It specifies the position and spacing of one line in a rectangular mesh.

LINE

  { X | Y } LOCATION= <n> [SPACING=<n>]

X | Y   Specifies the x or y co-ordinate of the grid line specified by LOCATION
                  and of local grid spacing given by SPACING.

2.3.3 SAVEFILE

This statement writes mesh and solution information into a file. By default, a structure file is
created. These structure files can be read with the **INITIALIZE** or **LOADFILE** statements.

SAVEFILE

  OUT.FILE= <c>

OUT.FILE is the identifier for the structure file to be written.

2.3.4. DEPOSITION

Material is deposited on the exposed surface of the structure, with the upper surface of the

deposited layer becoming the new exposed surface.

DEPOSITION

        { MATERIAL=<c> | SILICON | OXIDE | OXYNITRI | NITRIDE | POLYSILI |
ALUMINIUM | (PHOTORES [{ POSITIVE | NEGATIVE }])
        }

[IMPURITY=<c>] [ANTIMONY =<n> | ARSENIC=<n> | BORON=<n> | PHOSPHOR=<n>
]   THICKNES=<n>

It deposits one of the given MATERIALS, IMPURITIES of specified concentration and
THICKNES specifies the thickness of the deposited layer.

2.3.5. ETCH

It is used to remove a portion of the current structure. The user defines the region, material to be
removed. By default, all materials are etchable.
ETCH

 [{ MATERIAL=<c> | SILICON | OXIDE | OXYNITRI | NITRIDE | POLYSILI | PHOTORES
| ALUMINIUM }]

{ THICKNES=<n> (START | CONTINUE | DONE } X=<n> Y=<n>) | ISOTROPI | ALL }

MATERIAL specifies one of the given materials to be etched. START gives the first,
CONTINUE gives the next and DONE gives the last point in a series defining the region to be
etched, whose X and Y co-ordinates have been specified.

ISOTROPI removes material within the THICKNES of an exposed surface. ALL etches the
entire material that is specified.

2.3.6. IMPLANT

 It is used to simulate the ion implantation of impurities into the structure.

IMPLANT

   DOSE=<n> ENERGY=<n> { IMPURITY=<c> | ANTIMONY | ARSENIC | BORON | BF2 |
PHOSPHOR }

IMPURITY specifies one of the given materials to be implanted, of given DOSE (atoms/cm^2)
and ENERGY (KeV) of implant beam.

2.3.7. DIFFUSION

The DIFFUSION statement is used to model high temperature diffusion in both oxidising and non oxidising environments.

DIFFUSION

TIME=<n> [CONTINUE] TEMPERAT=<n> [{ T.RATE=<n> | T.FINAL=<n> }]
[{ DRYO2 | WETO2 | STEAM | INERT | AMB.1 | AMB.2 | AMB.3 | AMB.4 | AMB.5 | ([
F>O2=<n>] [F.H2O=<n>] [F.H2=<n>] [F.N2=<n>] [F.HCL=<n>]) }]

[IMPURITY=<c> [I.CONC=<n>] [ANTIMONY=<n>]   [ARSENIC=<n>]   [BORON=<n>]
[PHOSPHOR=<n>]
[PRESSURE=<n>] [{ P.RATE=<n> | P.FINAL=<n> }] [HCL=<n>]

TIME specifies the duration in mins. CONTINUE indicates that this step is a continuation of a previous diffusion step. Ambient temperature at the beginning(TEMPERAT) , the rate of change of temperature (T.RATE) and at the end of the step (T.FINAL) are specified in degrees celsius. Ambient gas(es) and their associated flows are specified as given.
IMPURITIES and their CONCENTRATION in the ambient gas at the surface of the structure can be specified. The total PRESSURE at the start of the step, time rate of change of pressure (P.RATE), and final pressure at the end of the step, (T.FINAL) can also be specified as shown.

2.3.8. EPITAXY

It is used to grow silicon material on the exposed silicon surface of the current structure.

EPITAXY

TIME=<n> TEMPERAT=<n> [{ T.RATE=<n> | T.FINAL=<n> }]
[IMPURITY=<c>   [ANTIMONY=<n>]   [ARSENIC=<n>]   [BORON=<n>]   [
PHOSPHOR=<n>]

THICKNESS=<n>

TIME specifies the duration of the step in mins. The temperature at the beginning of the step (TEMPERAT), it's time rate of change (T.RATE), and the final temperature (T.FINAL) are specified in degrees Celsius. IMPURITIES and their concentration can also be specified.
THICKNES of the deposited layer can also be specified.

2.3.9. SELECT

It is used to evaluate a quantity to be printed or plotted. It is also used to specify plot titles and axis labels.

SELECT

[Z=<n>] [TEMPERAT=<n>] [LABEL=<n>] [TITLE=<n>]

Z gives the expression defining the quantity to be printed or plotted at the given TEMPERAT. LABEL on the y-axis of a 1-D plot or z-axis of a 3-D plot and the TITLE can also be specified.

2.3.10. PRINT.1D

{ X.VALUE=<n> | Y.VALUE=<n> | ({ MATERIAL=<n> | SILICON | OXIDE | OXYNITRI | NITRIDE | PLOYSILI | PHOTORES | ALUMINUM }

{ /MATERIA=<n> | /SILICON | /OXIDE | /OXYNITR | /NITRIDE | /PLOYSIL | /PHOTORE | /ALUMINU | /AMBIENT | /REFLECT }) }

[LAYERS] [X.MIN=<n>] [Y.MIN=<n>]

The *x* co-ordinate of the vertical section and the *y* co-ordinate of the horizontal section along which values are to be printed in microns, are given by X.VALUE and Y.VALUE.
MATERIAL, prints values in the named material, at the interface with the other specified material.
/MATERIA prints values in the other specified material, at the interface with the named material.
LAYERS reports the integral of the specified quantity over each layer of the device.
X.MIN and X.MAX give the minimum and maximum position along the cross-section to be printed.

2.3.11. PLOT.1D

Plots the value of the selected expression along a line through the structure.

PLOT.1D

 { { [{ X.VALUE=<n> | Y.VALUE=<n> }] | ({ MATERIAL=<C> | SILICON | OXIDE | OXYNITRI | NITRIDE | POLYSILI | PHOTORES | ALUMINUM }

{ /MATERIA=<c> | /SILICON | /OXIDE | /OXYTNITR | /NITRIDE | /PLOYSIL | /PHOTORE | /ALUMINU | / AMBIENT | /REFLECT }) }

| { IN.FILE=<c> } [BOUNDARY] [CLEAR] [AXES] [CURVE] [LINE.TYP=<n>] [COLOR=<n>] [LEFT=<n>] [RIGHT=<n>] [BOTTOM=<n>] [TOP=<n>]
[X.OFFSET=<n>] [X.LENGTH=<n>] [X.SIZE=<n>]
[Y.OFFSET=<n>] [Y.LENGTH=<n>] [Y.SIZE=<n>]

X.VALUE and Y.VALUE give the position where a vertical or a horizontal cross-section is to be plotted, respectively.
MATERIAL plots the cross-section through the named material, at the interface of the other specified material.

/MATERIA plots a cross-section through the other specified material, at the interface with the named material.

IN.FILE contains the data to plot. BOUNDARIES that are crossed can be indicated by dashed vertical lines using this parameter.

Graphics screen is cleared (CLEAR) before drawing. AXES gives scaling information for drawing axes.

CURVE specifies that a line is to be drawn through the data points. LINE.TYP indicates the dashed line type and COLOR indicates the color used for plotting data.

The minimum and maximum values to be plotted on *x* and *y* axes are specified by LEFT, RIGHT, BOTTOM and TOP, respectively. Also, the OFFSET, LENGTH and SIZE of characters in both axes can be specified as given.

2.3.12. PLOT.2D

This statement specifies a two-dimensional plot of the device structure.

PLOT.2D

  [X.MIN=<n>] [X.MAX=<n>] [Y.MIN=<n>] [Y.MAX=<n>] [SCALE]
  [CLEAR] [AXES] [BOUNDARY] [L.BOUND=<n>] [C.BOUND=<n>]
  [GRID] [L.GRID=<n>] [C.GRID=<n>]
  [[STRESS] [FLOW] VLENG=<n> [VMAX=<n>]
    [L.COMPRE=<n>] [C.COMPRE=<n>] [L.TENSIO=<n>] [C.TENSIO=<n>]]

The minimum and maximum *x* and *y* co-ordinates and scale, to preserve the true aspect ratio of the device are specified as given. CLEAR is used to clear the screen before the graph is drawn. AXES statement is used to draw axes according to specified scale. The line type and color of device boundary and material interface are specified by L.BOUND and C.BOUND and that of the grid, by L.GRID and C.GRID, respectively.

STRESS plots the principal stresses in the structure. FLOW indicates the velocity (due to oxidation) or displacement (using STRESS) at each mesh point. The length and maximum value of stress vector are specified as given.

The line type and color of tensile and compressive stresses can also be specified.

2.3.13. LABEL

The label statement is used to add labels to the plot.

LABEL

  (X=<n> Y=<n> [CM]) [SIZE=<n>] [COLOR=<n>]
  [LABEL=<c> [{ LEFT | CENTER | RIGHT }]]

[LINE.TYP=<n>] [C.LINE=<n>] [LENGTH=<n>]

The *x* and *y* location of the LABEL's position (LEFT, RIGHT, CENTER) and it's units (c.m) is specified as given
The SIZE of the characters and COLOR of the text are also specified. The LINE.TYP, color (C.LINE) and LENGTH of the line segment to be drawn before the label is specified.

CHAPTER 3

Example #1:

This example illustrates the diffusion of a dopant onto a n-type substrate and the resulting plot.
To add the second plot to the first one, we use the ^Clear and ^Axes specifications to prevent the clearing of the screen and the drawing of a new axes, respectively.

CODE FOR FIRST TEST.INP

$ FIRST PROG!

$ Automatic grid generation for the initial substrate
INITIALIZE PHOSPHORUS=2E15

$ Diffuse Boron and drive in
DIFFUSION TEMP=950 TIME=30 BORON=2E20
DIFFUSION TEMP=1050 TIME=90

$ Diffuse Phosphorus
DIFFUSION TEMP=1050 TIME=10 PHOSPHORUS=1E21

$ Plot Results
SELECT Z=LOG10(BORON) LABEL=LOG(CONCENTRATION)
PLOT.1D BOTTOM=14 TOP=21 RIGHT=4 LINE.TYP=5 COLOR=2
SELECT Z=LOG10(PHOSPHORUS)
PLOT.1D ^AXES ^CLEAR LINE.TYP=2 COLOR=3

OUTPUT OF TEST.INP

***************** Advanced Application Module Availability
Extended Defects   : not available
User-Specified Equation Interface   : not available
***

 Entering source file test.inp.

$ FIRST PROG!

INITIALIZE PHOSPHORUS=2E15
** Automatic X grid generation: lines at X=0 and X=1 micron.
** Automatic Y grid generation.
   2 lines in the x direction.
   36 lines in the y direction.

DIFFUSION TEMP=950 TIME=30 BORON=2E20
DIFFUSION TEMP=1050 TIME=90
DIFFUSION TEMP=1050 TIME=10 PHOSPHORUS=1E21
SELECT Z=LOG10(BORON) LABEL=LOG(CONCENTRATION)
PLOT.1D BOTTOM=14 TOP=21 RIGHT=4 LINE.TYP=5 COLOR=2
** Plotting along X.VALUE=0.

** Plotting to device: "x" (specified by DEFPDEV environment variable)
SELECT Z=LOG10(PHOSPHORUS)
PLOT.1D ^AXES ^CLEAR LINE.TYP=2 COLOR=3
** Plotting along X.VALUE=0.

****
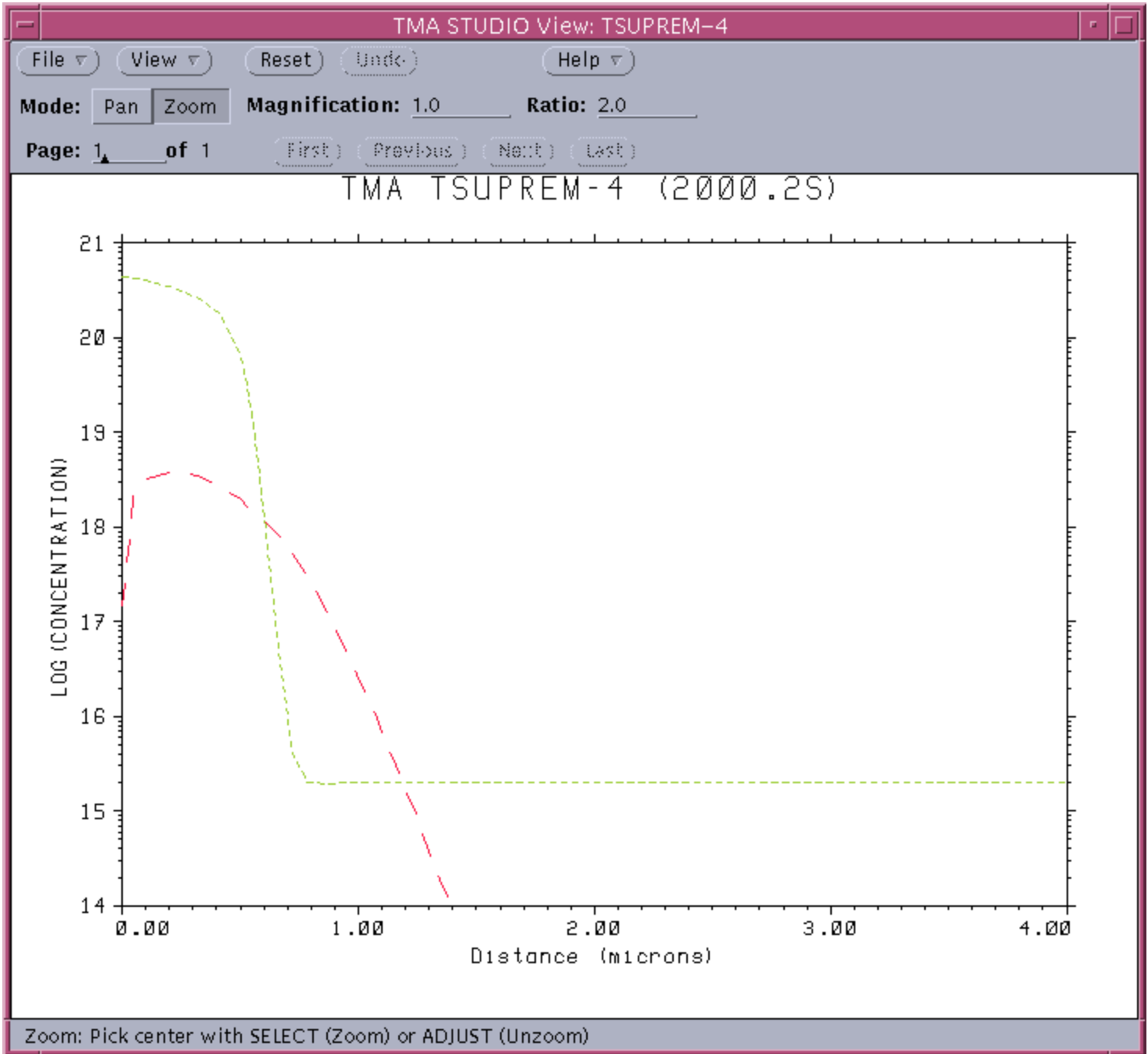
Example #2:

This example presents a one-dimensional simulation of an oxide isolated bipolar transistor structure, from the buried collector region up through the emitter.
The initial substrate is doped with Boron and oxide grown over it to mask the buried layer implant. Oxide from the active region is etched and buried layer is implanted and driven in. Again oxide is removed with the etch statement. Next, epitaxial layer is grown. Finally, pad oxide is grown and nitride mask is deposited for field-isolation oxidation.
The resulting structure is saved in the output file named EX1A and can be used as a starting point for the remaining process steps. The results are plotted as shown.

Code for EXIA.inp file


```
$ TSuprem4 ex 1(a)
$ Bipolar Active Device Region: Buried Layer and epitaxial deposition

$ Use automatic grid generation and adaptive grid
INITIALIZE BORON=1E15

$ Grow buried layer masking oxide
DIFFUSION TEMP=1150 TIME=120 STEAM

$ Etch the buried layer masking oxide
ETCH OXIDE ALL

$ Implant and drive in the antimony buried layer
IMPLANT ANTIMONY DOSE=1E15 ENERGY=75
DIFFUSION TEMP=1150 TIME=30 DRYO2
DIFFUSION TEMP=1150 TIME=360

$ Etch the oxide
ETCH OXIDE ALL

$ Grow 1.8 micron of arsenic-doped epitaxy
EPITAXY THICKNESS=1.8 SPACES=9 TEMP=1050 TIME=6 ARSENIC=5E15
```

```
$ Grow pad oxide and deposit nitride
DIFFUSION TEMP=1050 TIME=30 DRYO2
DEPOSITION NITRIDE THICKNESS=0.12

$ Save initial active region results
SAVEFILE OUT.FILE=EX1A

$ PLot results
SELECT Z=LOG10(BORON) TITLE="Active, Epitaxy" LABEL=LOG(CONCENTRATION)
PLOT.1D BOTTOM=13 TOP=21 RIGHT=5 LINE.TYP=2 COLOR=2
SELECT Z=LOG10(ARSENIC)
PLOT.1D ^AXES ^CLEAR LINE.TYP=2 COLOR=3
SELECT Z=LOG10(ANTIMONY)
PLOT.1D ^AX ^CL LINE.TYP=3 COLOR=3

$ Label plot
LABEL X=4.2 Y=15.1 LABEL=Boron
LABEL X=-.8 Y=15.8 LABEL=Arsenic
LABEL X=2.1 Y=18.2 LABEL=Antimony

$ Print Layer information
SELECT Z=DOPING
PRINT.1D LAYERS
                    *********************
```
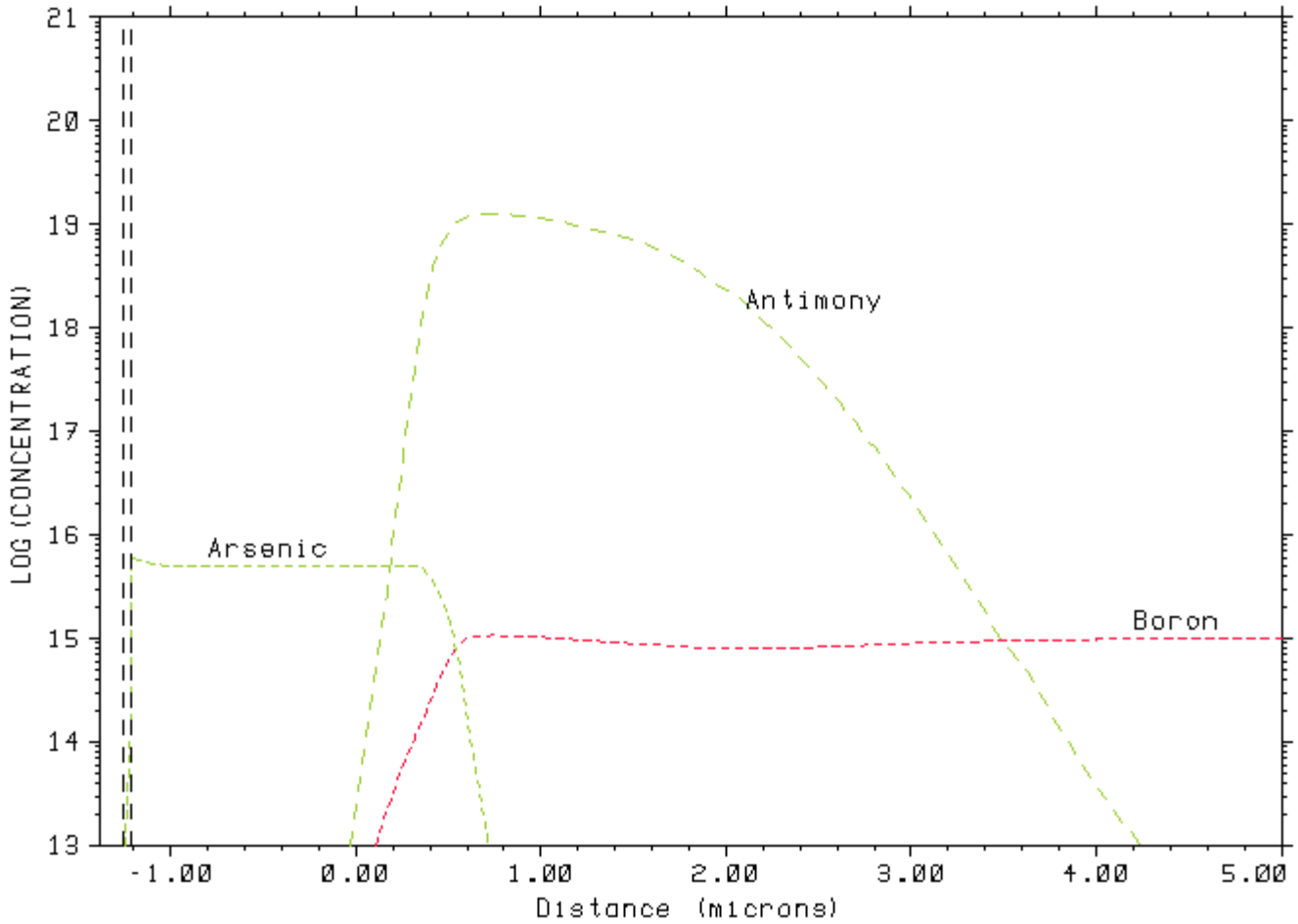
File ▽   View ▽   Reset   Undo   Help ▽

Mode: Pan Zoom   **Magnification:** 1.0   **Ratio:** 2.0

Page: 1xv▾ **of** 1   First   Previous   Next   Last

Active, Epitaxy

Plot of LOG(CONCENTRATION) versus Distance (microns), showing curves labeled Antimony, Arsenic, and Boron.

Example #3:

This example presents a two-dimensional simulation of a local oxidation process.

First a mesh is defined, followed by pad-oxide deposition. To mask the oxidation, a nitride layer is grown and patterned using 'etch' statement. An oxidation model is used to perform oxidation. The final structure is saved in the file called LOCOS.

A two-dimensional plot is used to plot the structure boundaries and material interfaces. Material flow in the structure at the end of oxidation is shown using the FLOW statement. Also, stresses in the structure are plotted. Continuous lines indicate compressive stresses and dashed lines indicate tensile stress. As expected, the stresses are concentrated near the portion of the structure where the oxide growth is non-uniform and where the nitride is forced to bead.

CODE FOR LOCOS.INP

```
$ Narrow window example
$ Oxide Shape

$ Setup the grid
LINE X LOC=0.0 SPAC=0.15
LINE X LOC=1.25 SPAC=0.04
LINE X LOC=1.5 SPAC=0.1

LINE Y LOC=0 SPAC=0.03
LINE Y LOC=0.5 SPAC=0.1

$ No impurities for faster oxidation simulation
INITIALIZE

$ Deposit pad oxide and define nitride mask
DEPOSITION OXIDE THICKNESS=0.03 SPACES=2
DEPOSITION NITRIDE THICKNESS=0.10 SPACES=2
ETCH NITRIDE RIGHT P1.X=1.25

$ Plot the grid
SELECT TITLE="Grid for Oxidation"
PLOT.2D GRID SCALE C.GRID=2

$ Do the oxidation
METHOD VISCOEL DY.OXIDE=0.05 INIT=0.15
MATERIAL MAT=OXIDE VC=425
MATERIAL MAT=NITRIDE VC=170
```
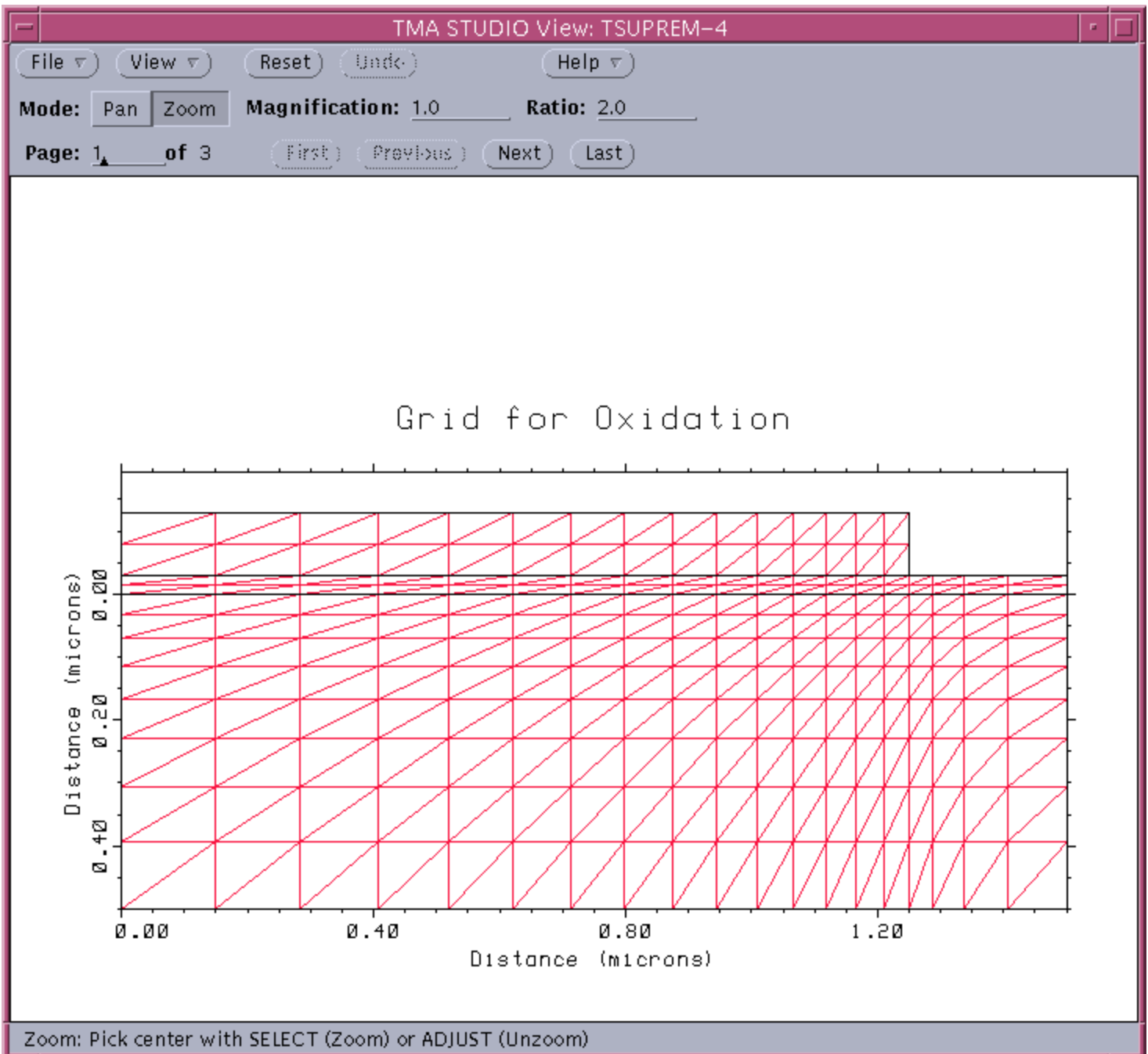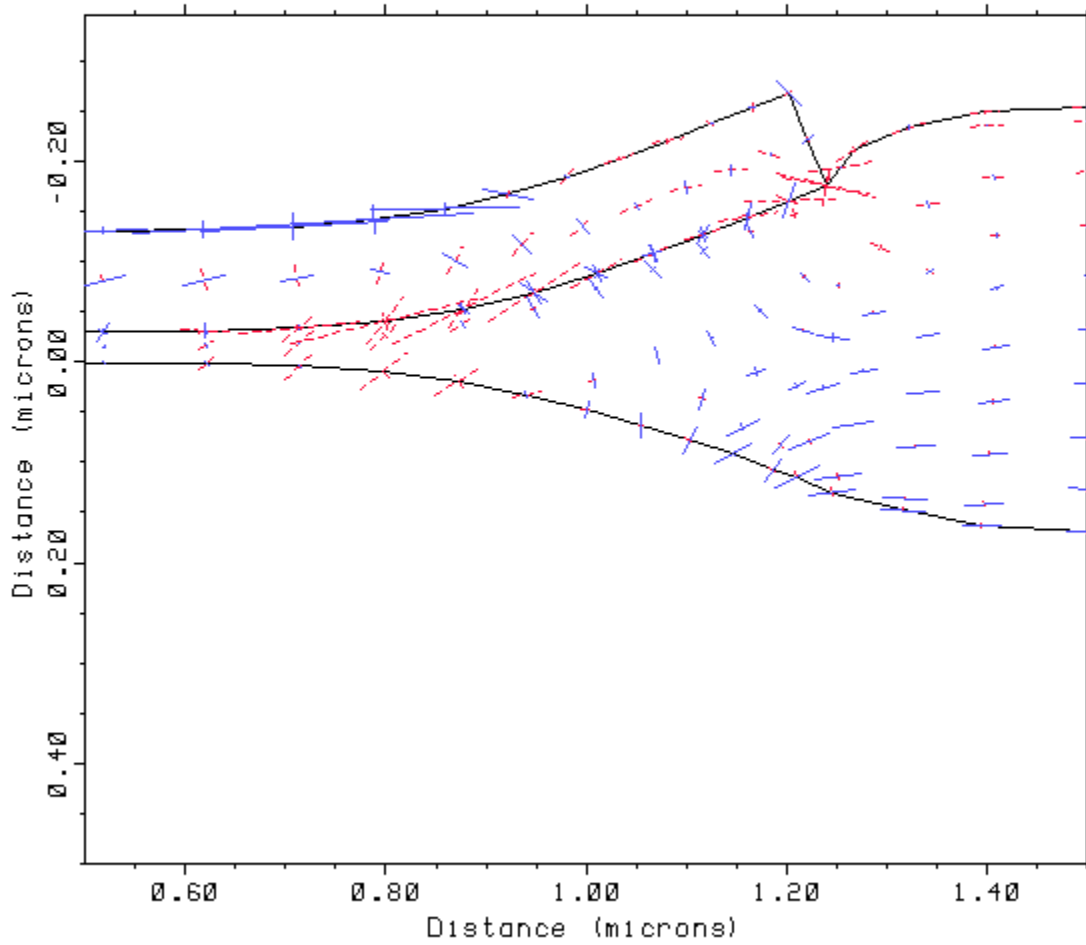
DIFFUSION TEMP=1000 TIME=100 WETO2

$ Save the structure
SAVEFILE OUT.FILE=LOCOS

$ Plot the structure, showing flow lines
SELECT TITLE="Flow at the end of oxidation step"
PLOT.2D SCALE FLOW VLENG=0.065 X.MIN=0.5 C.FLOW=4

$ Plot the final structure, showing stress vectors
SELECT TITLE="Stresses after oxidation"
PLOT.2D SCALE STRESS VLENG=0.2 X.MIN=0.5 +
        C.COMPRE=4 C.TENSIO=2 L.TENSIO=2
                    ****

TMA STUDIO View: TSUPREM-4

File ▽   View ▽   Reset   Undo   Help ▽

Mode: Pan | Zoom   Magnification: 1.0   Ratio: 2.0

Page: 1 of 3   First   Previous   Next   Last

Grid for Oxidation

Zoom: Pick center with SELECT (Zoom) or ADJUST (Unzoom)

Stresses after oxidation

Flow at the end of oxidation step

Distance (microns)

Distance (microns)