Scientific Forming Technologies Corporation

2D MACHINING (CUTTING) LAB

Problem Summary:

This lab will briefly demonstrate how to use the machining template to prepare model data representing orthogonal cutting conditions (cutting edge is orthogonal/perpendicular to the cutting direction) to simulate chip formation process. Interactive definition of the process conditions, cutting edge, coating, material, steady state and tool stress related features are discussed here. Brief description of the orthogonal cutting conditions and how the process parameters are related to the modeling domain are indicated as shown in the **Figure 1**.

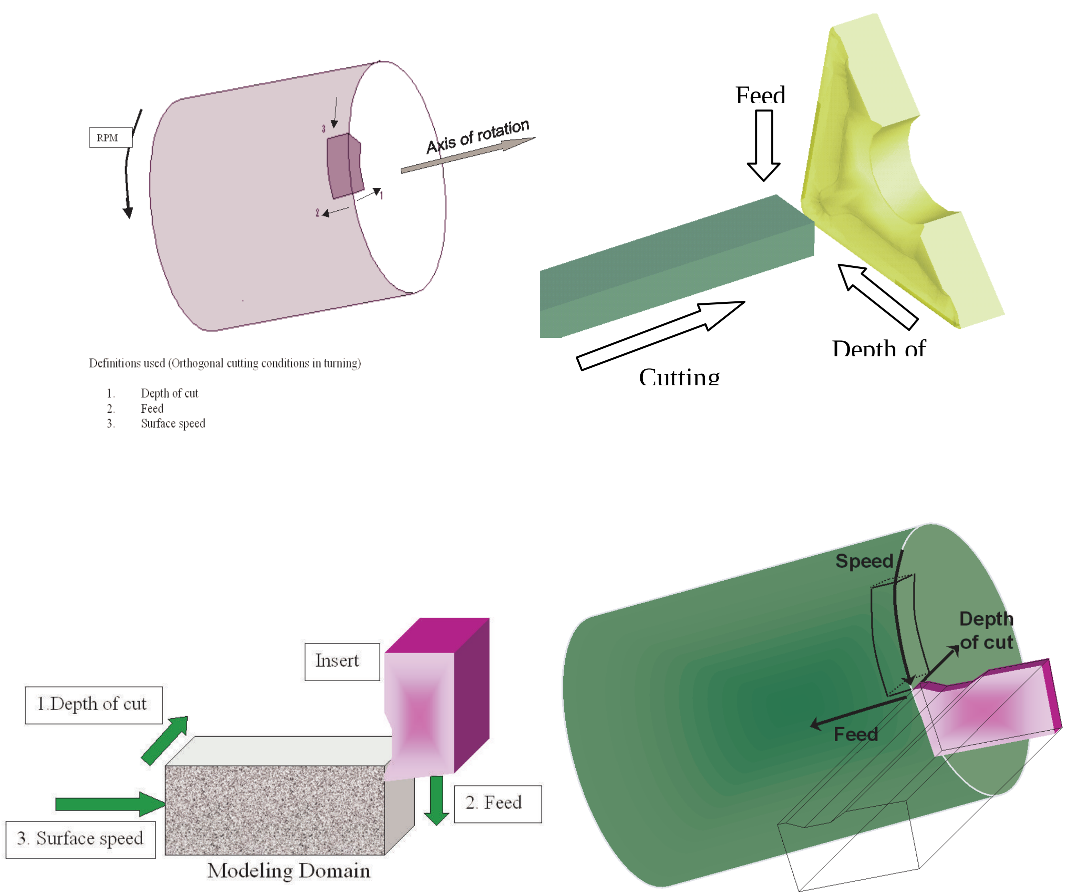

Figure 1: Relationship between the process data and analysis domain

1. Starting the System

Open the Machining 2D template from the program installation menu (All programs) on PC, and using the alias ‘2d_cutting’ on Unix to access complete features of the template that include problem setup, steady state and tool stress analysis. The other way activating this template from the Deform 2D main menu allows user to use the problem setup part only.

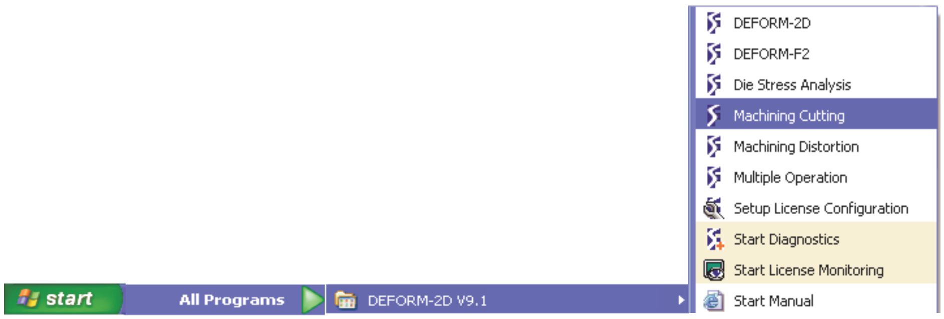

Figure 2: Starting up the system

2. Set Unit System

Open a new project (top left most icon). Enter project name as “MACHINING” and title as “2D_CUTTING”. Set the units system to SI (**Figure3**) for this lab and click on ‘next’. Enter the operation name as ‘2d_cutting’ and click on ‘next’.

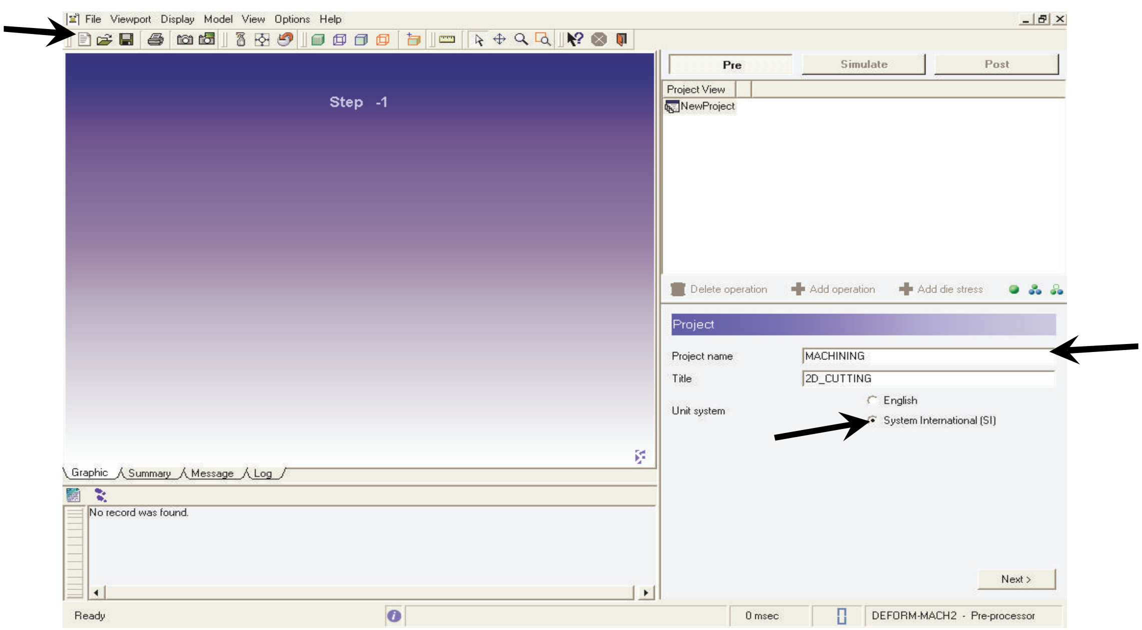

Figure 3: Unit system set up

3. Defining Machining Process Data

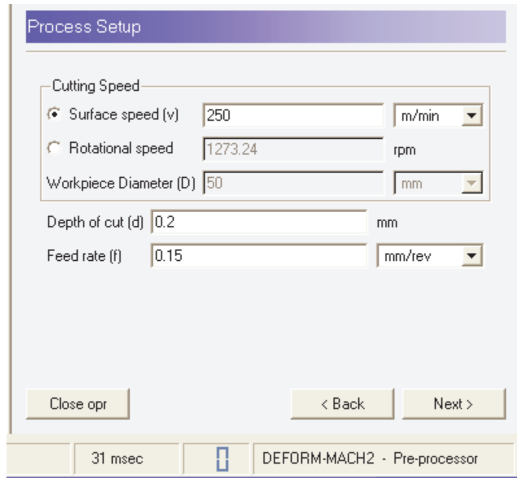

Figure 4: Project setup window

In the 'Process Setup' menu, select the 'surface speed as 250 m/min, depth of cut as 0.2 mm, and feed rate as 0.15 mm/rev' (**Figure 4**). Alternatively user can also specify rotational speed of the work piece and part diameter instead of surface speed.

4. Defining Process Environment Conditions

Under the 'Process Condition' menu, enter the environment temperature as 22.5 C, shear friction factor as 0.48 (between the chip and the insert) and the interface heat transfer coefficient as 39.5 N/sec/mm/C. Click 'Next' to continue. **(Figure 5).**

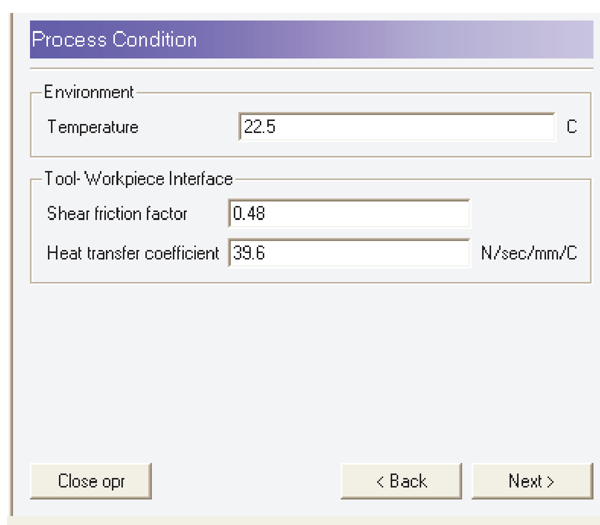

Figure 5: Process Setup Menu

5. Defining Cutting Edge Geometry

In the ‘Tool Setup’ menu, enter the tool temperature. At this point user can import the cutting edge geometry from a previously defined simulation keyword file or a database file. For this lab we define the cutting edge geometry using the primitives available with the system. Click ‘Next’ to enter the ‘Insert Geometry’ menu **(Figure 6)** and select ‘Define Primitive Geometry’ to enter ‘Geometry Primitives menu’. Select option ‘Insert 3’ to define the parameters for a generic cutting edge details.

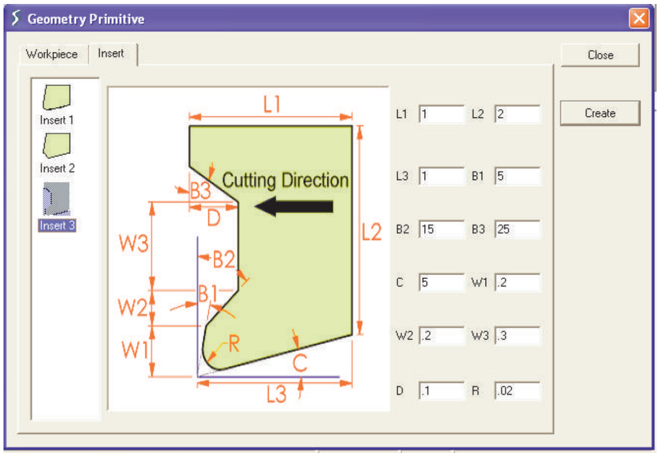

Figure 6: Geometry Definition Window

Use the parameters as follows:

L1 = 1.0, L2 = 2.0, L3 = 1.0, B1 = 5 deg., B2 = 15 deg., B3 = 25 deg., C = 5 deg., W1 = 0.2, W2 = 0.2, W3 = 0.3, D = 0.1, R = 0.02.

In the above set of parameters R represents hone radius of 20 microns. Click on ‘Create’ to display the corresponding geometry in the display area. Then click on ‘Close’ to come out of ‘Geometry Primitive’ menu and click ‘Next’ to proceed.

6. Coating Definition

In the subsequent ‘Create Coating Mesh’ menu, add (+) two layers of coating details. For the outer layer select 10 microns as coating thickness and ‘TiN’ as the material and for the inner layer select 10 microns as the coating thickness and ‘Al2O3’ as the coating layer from tool _material. Click ‘Next’ to proceed. **(Figure 7)**

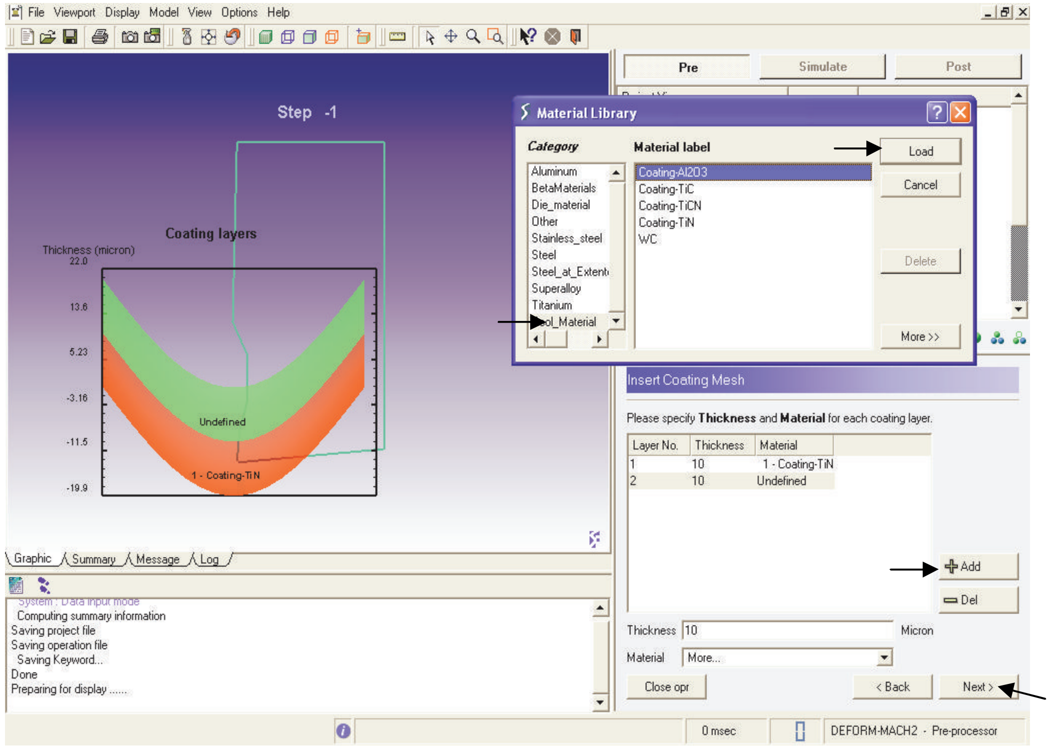

Figure 7: Defining Coating Material

In the Tool mesh generation menu select '700' as the total number of elements for the insert and click on 'Generate mesh' **(Figure 8)** . Program automatically generates the necessary tool mesh and displays the relative mesh gradients and windows used. Click 'Next' to proceed.

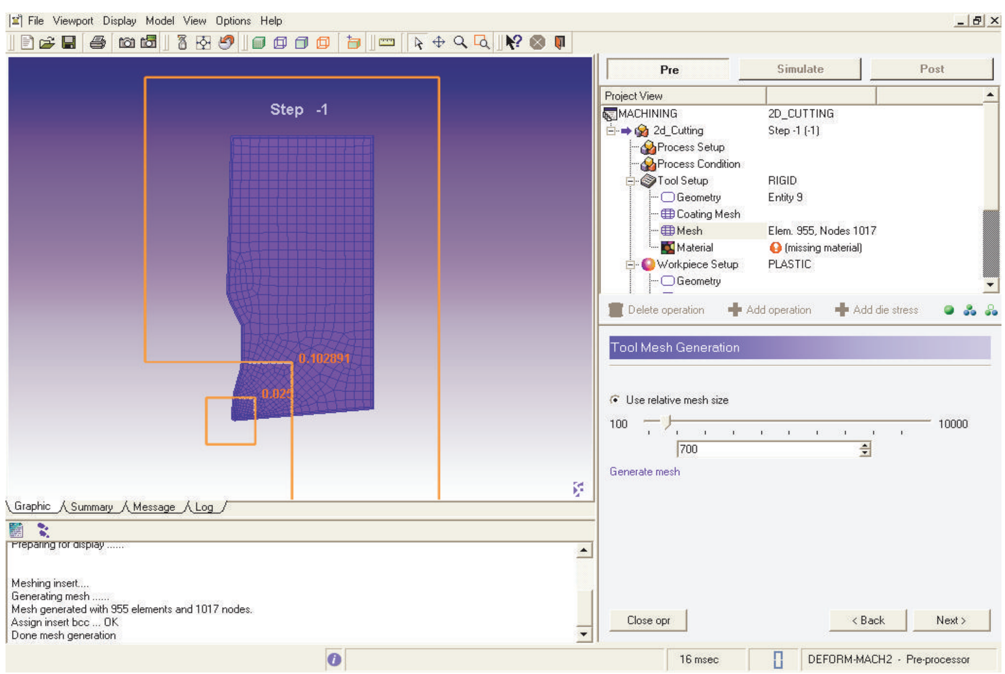

Figure 8: Tool Mesh generation window

7. Insert Boundary Conditions

Next step is to view the thermal boundary conditions generated by the system in the 'View Insert BCC' menu. The program generates heat exchange with environment (part of which comes from insert contact region) and the fixed nodal temperature BCC for the surfaces away from the cutting edge. Click 'Next' to proceed. (**Figure 9**).

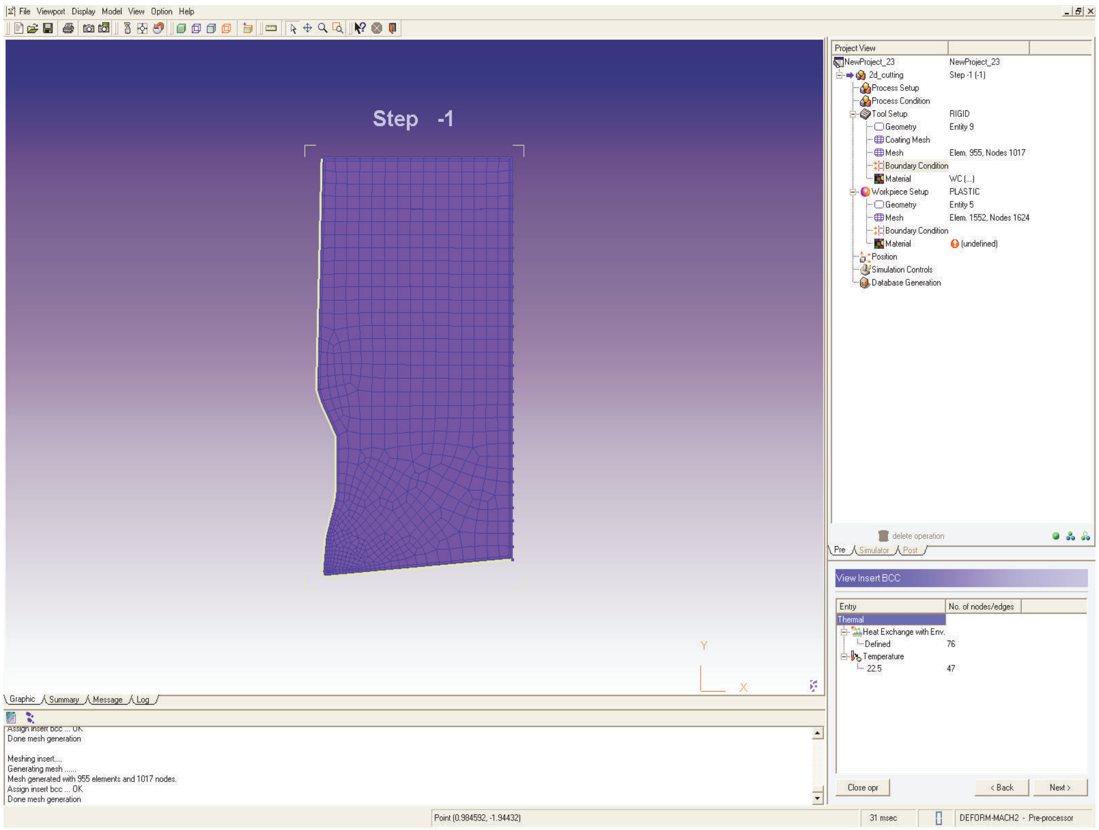

Figure 9: Boundary Condition for the tool

8. Select Insert Base Material

In the 'Insert Material Setup' menu, select the material from the library by selecting 'Import material from library'. Select 'Tool material' category and 'WC' from the resulting material set. Click 'Next' to proceed to the work piece setup.

9. Set the Work piece Geometry Data.

In the 'Work piece setup' define work piece as a plastic object and temperature as 22.5 C and click 'Next' to proceed.

In the 'Work piece Geometry' menu, select the option 'Define primitive geometry'. Using the parameters W = 3 and H = 1 'create' the work piece geometry. Close this geometry menu and click 'Next' to go to mesh generation page.

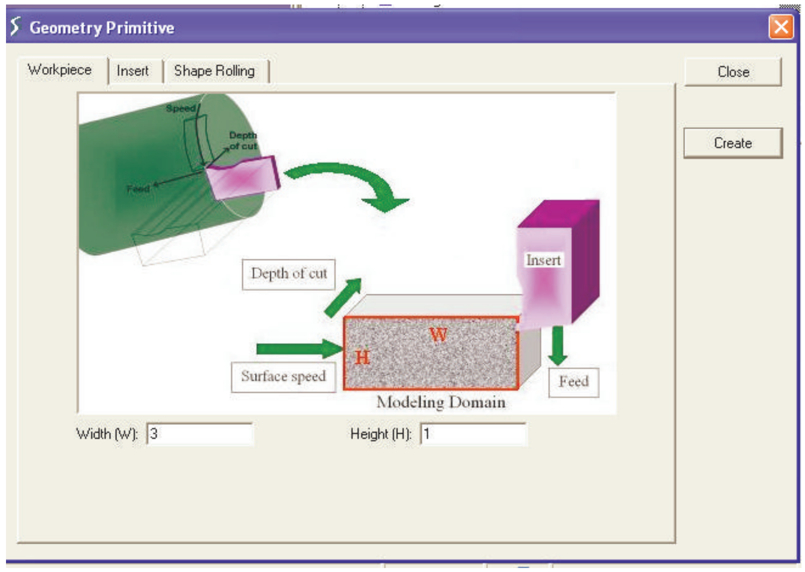

Figure 10: Defining workpiece geometry

Select 1500 elements for the work piece and click on ‘Generate mesh’ to complete mesh generation for the work piece. Click ‘Next’ to proceed.

10.Check the Work piece Boundary Conditions

In the ‘View Work piece BCC’ menu, check the boundary conditions imposed on work piece by the system, and click ‘Next’ to proceed.

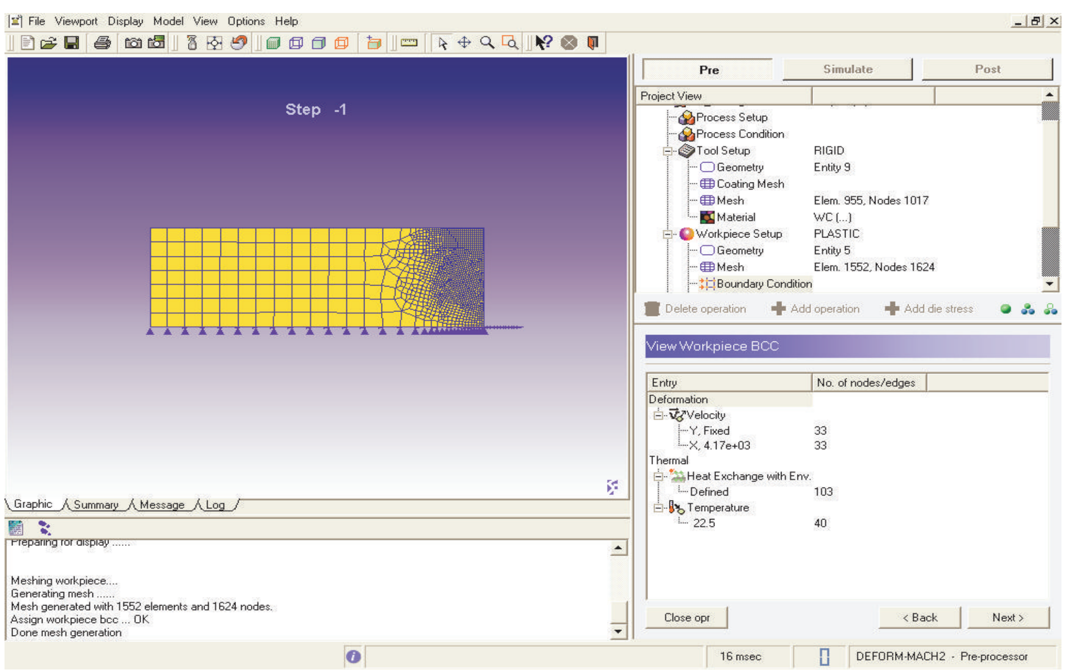

Figure 11: Boundary Condition for workpiece

11.Set the Work piece Material

In the ‘Work piece material setup’ menu, choose the option ‘import material from library’ option, select ‘Steel’ category and ‘AISI-1045 (machining)’. ‘Load’ this material and click ‘Next’ to proceed.

12.Insert Positioning

In the ‘Position’ menu, click on ‘Generate default position’ and click ‘Next’ to proceed.

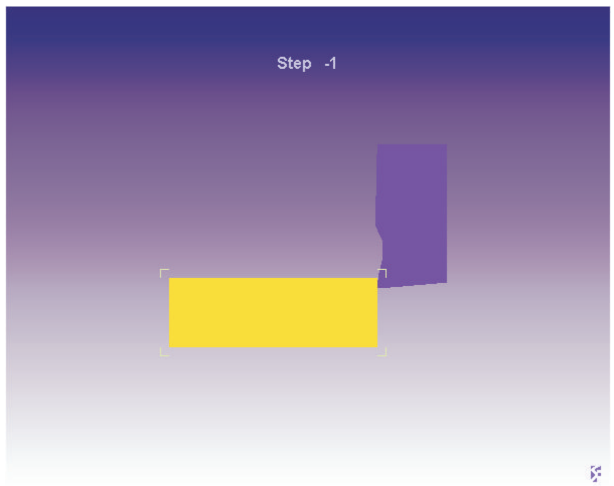

Figure 12: Objects after positioning

13.Tool Wear Setting

Select Usui’s model and input coefficients a , b, Please note that these (shown next to the equation) are only some typical values that can be obtained from the literature, and user is responsible for accuracy of this data. Click “Next”.

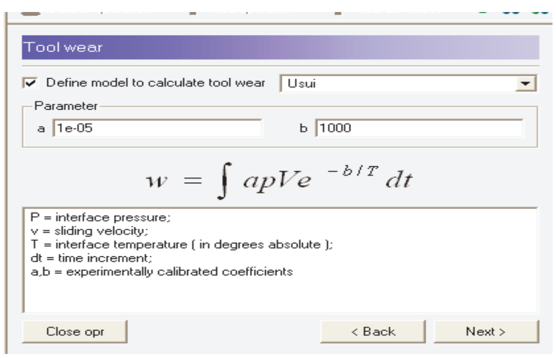

Figure 13: Tool wear setup window

14.Simulation Controls

Check the 'Simulation Controls' and set the length of cut as '1 mm' and leave the rest as default values and proceed to generate the Database. Close the current operation

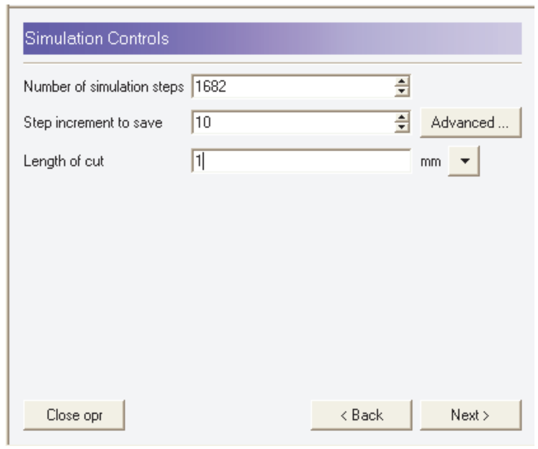

Figure 14: Simulation Control window

15.Running the Simulation

Select the 'Simulator' option

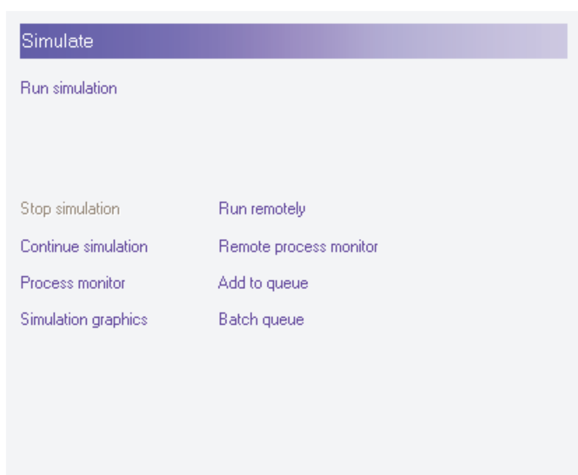

Figure 15: Running the simulation

Select 'Run Simulation' to start the simulation. User can use 'Simulation Graphics' to view the cutting process as the simulation proceeds to the specified cutting length.

After the simulation is complete, user can choose to verify the results in ‘Post’ or return to ‘Pre’ to add another operation to simulate steady state conditions or to add a tool stress analysis operation or to continue the current transient mode of chip formation.

We first check on the model setup for steady state analysis. Model setup for steady state analysis requires a prior transient analysis be completed resulting in sufficient chip formation and a clear contact region with tool.

16.Setting Data for Steady State Analysis

Select a step just prior to the last step from the transient analysis and enter ‘Pre’

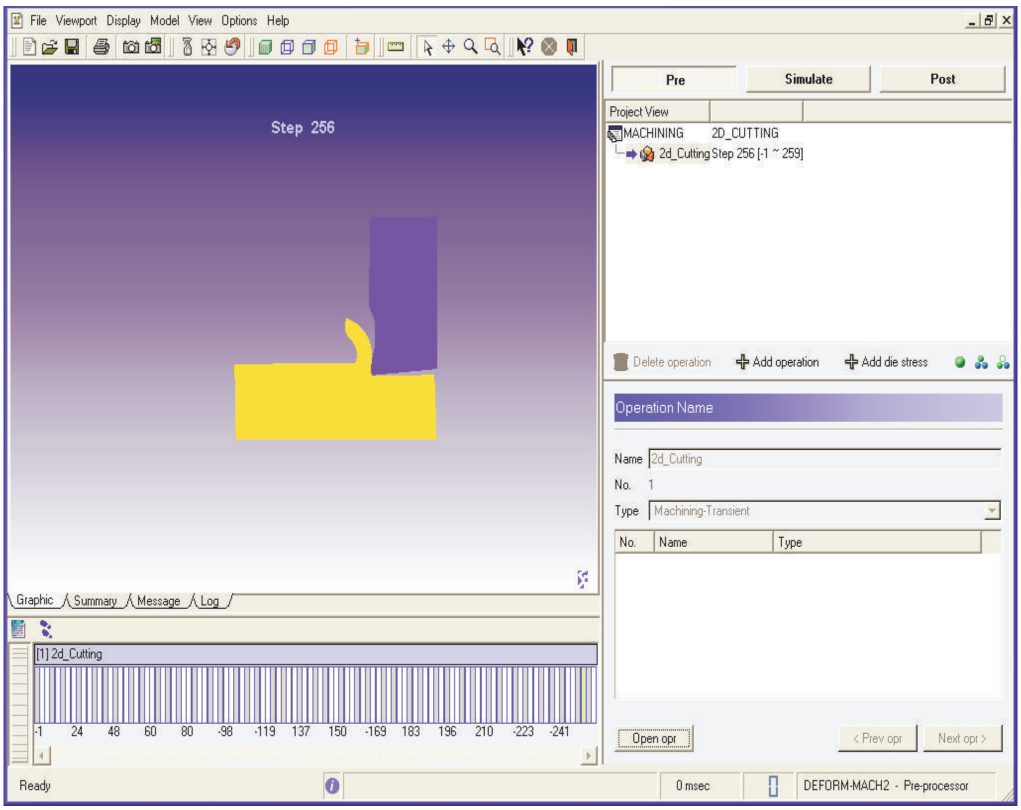

Figure 16: Selection Of Step for Steady State Analysis

Select ‘Add operation’ and select ‘Machining Steady State’ and select ‘Open Operation’ to define the necessary data. Click ‘Next’ to continue.

17.Identifying the End Surface of the Chip

Select Start and End points on the chip surface to define part of the exit surface for the material flow in steady state domain. Other material exit surface is far right surface of the work piece.

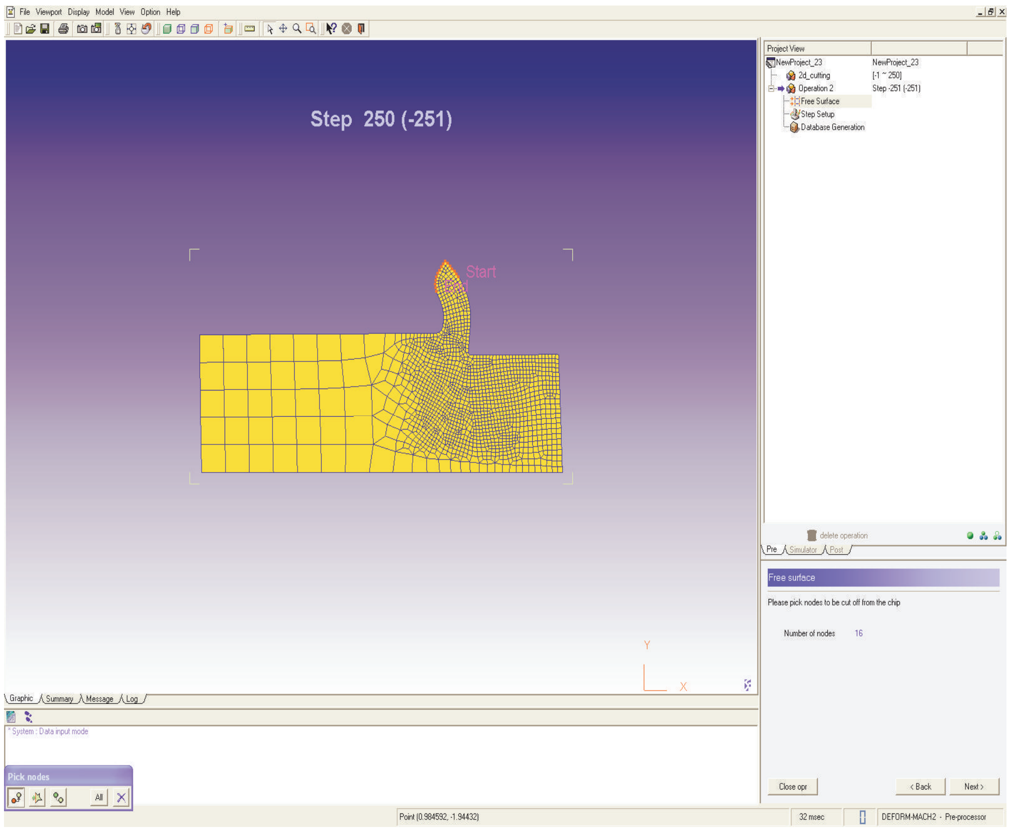

Figure 17: Identifying star and end point

Once start and end points are selected (and click on Next') on the chip surface template will remesh the remaining geometry.

18.Preparing for Steady State Run

Click 'Next' in the step setup menu and proceed to generate the database and close the operation. Go to the 'Simulator' and 'Run the simulation' Enter 'Post' to view the results. After the steady state run, proceed to add another transient run or Check the results in 'Post'.

19.Setting the Data for Tool Stress Analysis

Tool stress analysis can be added as an operation at any stage of chip formation. Typically after some chip formation user can choose to add an operation ‘Add die stress’ to carryout insert stress analysis. (select a positive step from the step list)

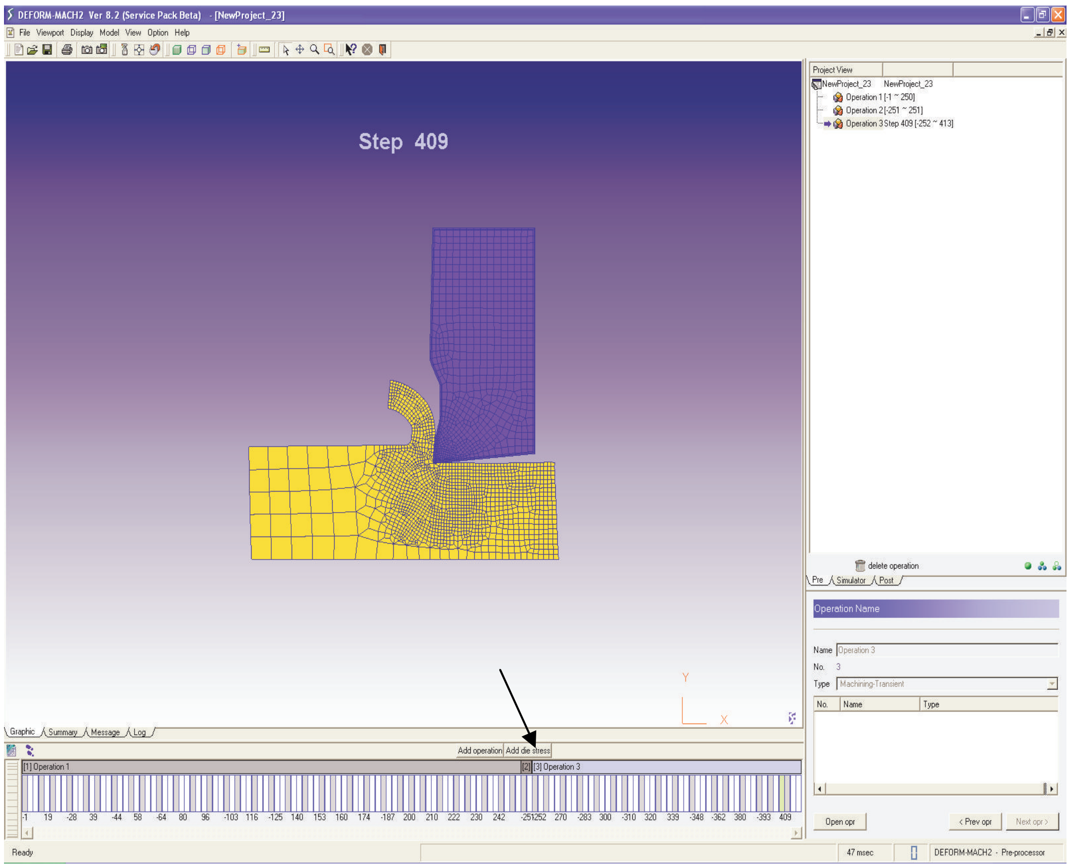

Figure 18: Tool stress analysis

Once a positive step is selected from the step list, click on ‘Add die stress’ to add a die stress operation. After setting operation name, click ‘Next’ to proceed.

20.Object Selection

Select the ‘Insert’ from the object selection menu and accept default ‘0’ in the ‘extra die components’ menu and click ‘Next’ to proceed. Set the simulation mode to ‘Isothermal’ and click ‘Next’

21.Tool Geometry and Mesh

Accept the default choice on tool selection, object type (leave as elastic) and existing mesh for the insert. Click ‘Next’ to proceed.

22.Interpolating the Cutting Forces on to the Insert

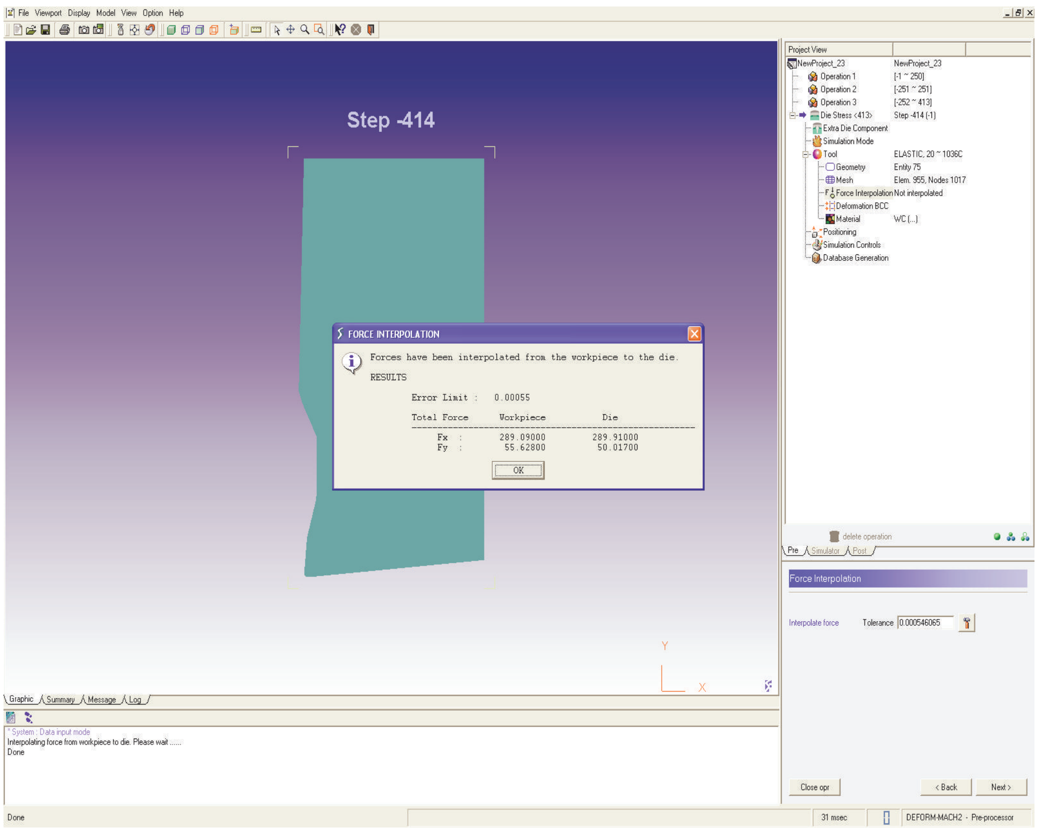

Figure 19: Interpolating forces

To interpolate the cutting forces on to the insert, click on ‘interpolate force’ using default tolerances. Click on ‘OK’ and ‘Next’ to proceed.

23.Deformation Boundary Conditions

Set the deformation boundary conditions for the insert, by selecting the (x+y) option, picking the starting and end point and (+) to confirm the selection. Click ‘Next’ to proceed.

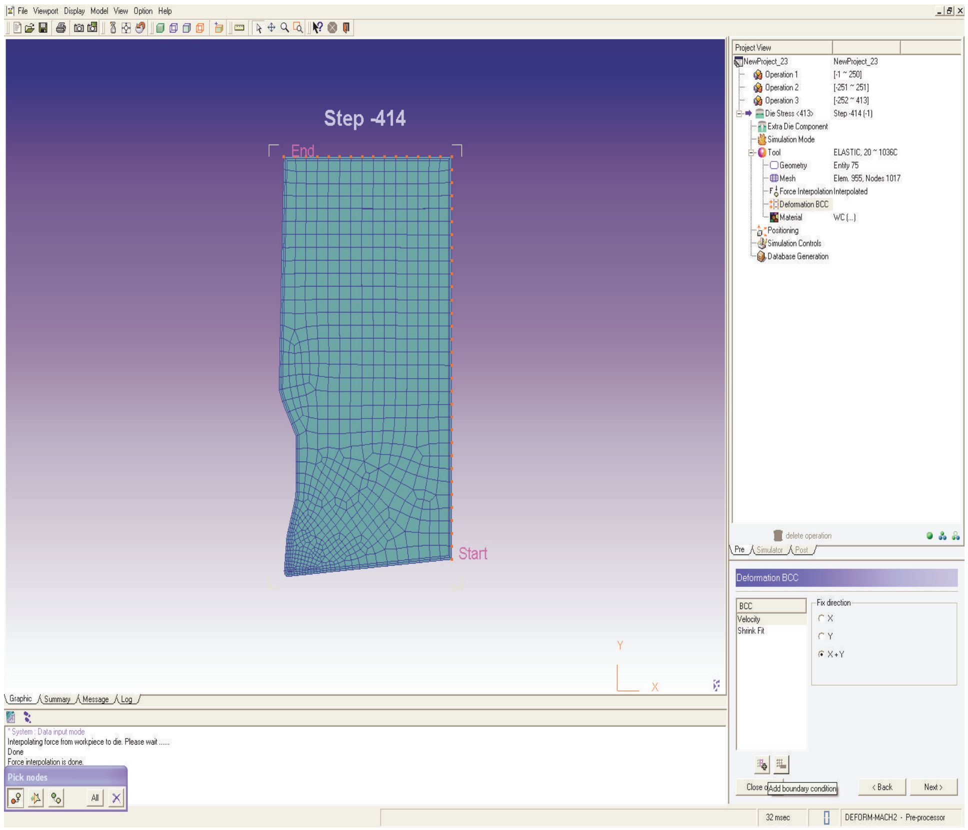

Figure 20: BCC applied for tool

24.Material Selection

Confirm the existing material selection, click ‘Next’ to proceed.

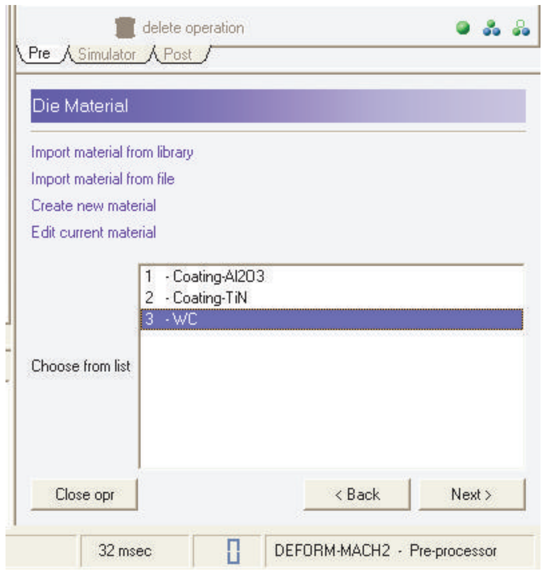

Figure 21: Material selection

25.Position and Simulation Controls

Accept the defaults for position and simulation control data and click ‘Next’. Generate the Database and proceed to close the operation, enter the ‘Simulator’ and run the simulation. After the simulation is run, enter ‘Post’ to view the results

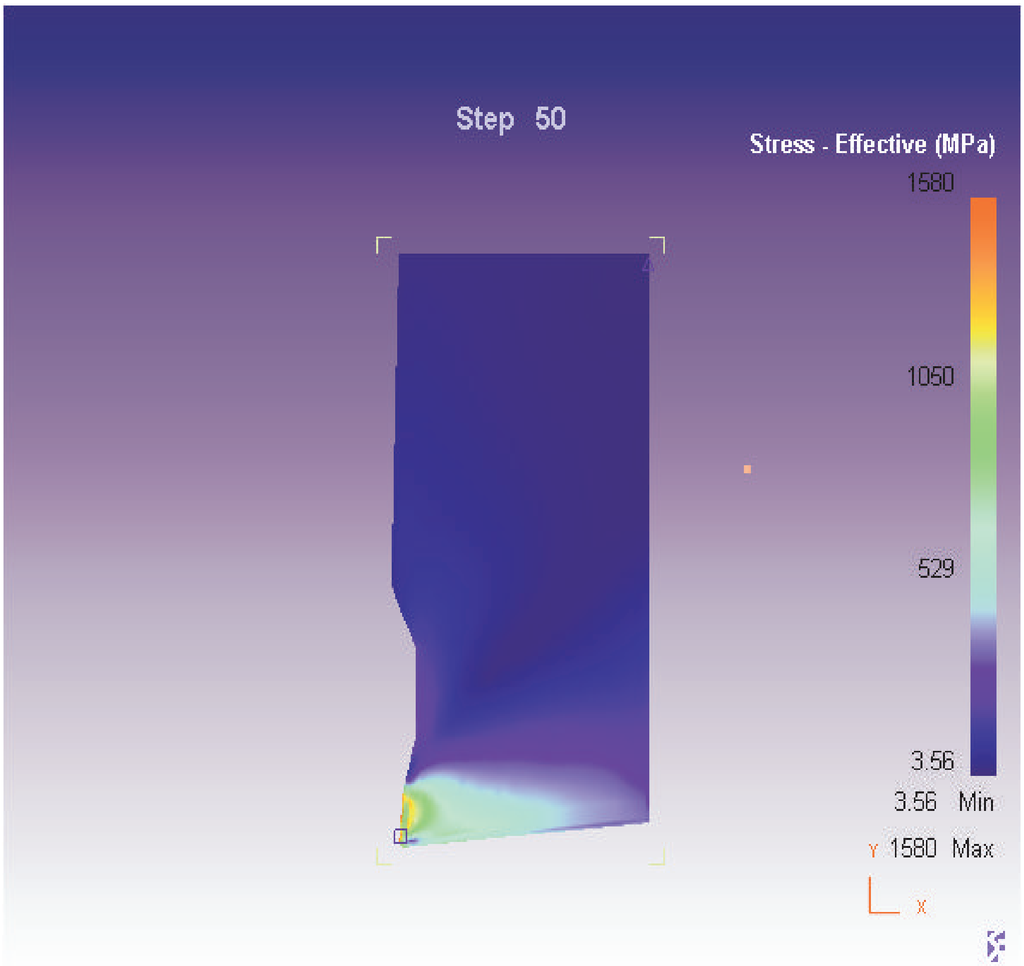

Figure 22: Stress - Effective

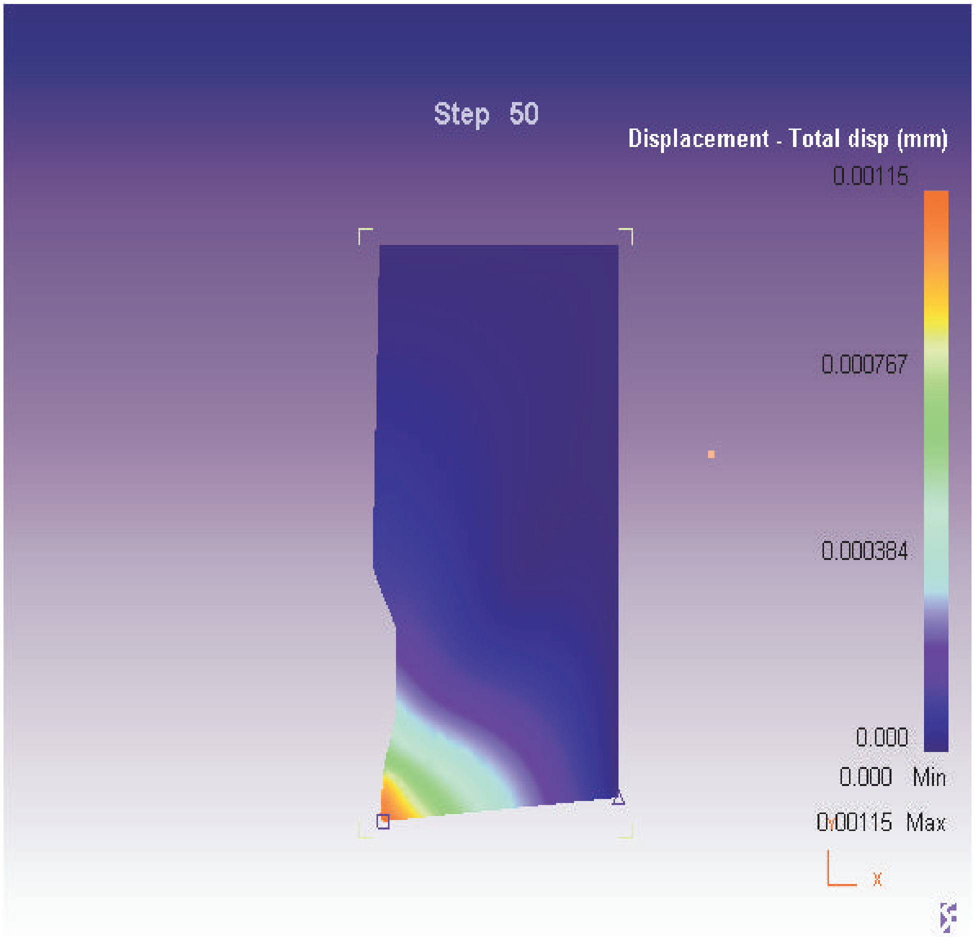

Figure 23: Total Displacement