# Dynaform-PC
Training Manual

Version 1.01

Introduction

Welcome to the DYNAFORM-PC Training Manual. This manual will take you through, step-by-step, the process of pre-processing and then post-processing a basic sheet metal forming simulation. In this manual we will be using a simple S-rail (NUMISHEET '96) as our test case.

I. Opening/Creating a DYNAFORM-PC Database and Analysis Setup

Start the Dynaform-PC pre-processor. The default path for Dynaform is C:\Program Files\Dynaform. In this directory, double click the executable file, **prepost.exe**. The pre-processor will be started.

After starting the Dynaform pre-processor, the **Open a Database File** dialog box is displayed:
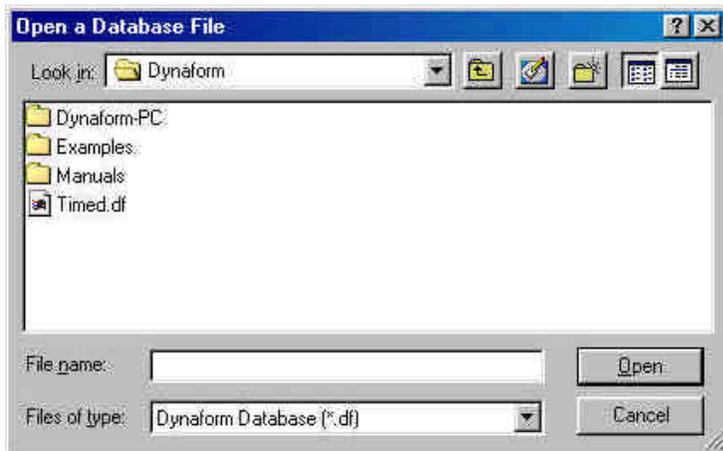
1. Since we have not yet created any databases that could be opened, press **Cancel**.
2. From the menu bar, select **File➜New**. The **Create a Database File** dialog box is displayed.
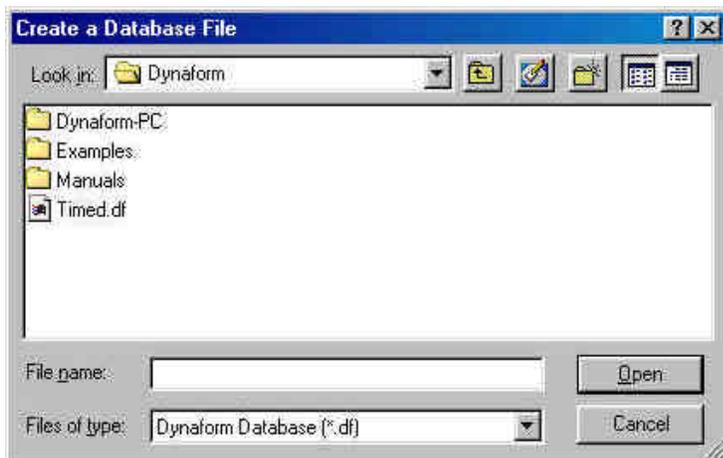
3. In this box, move to the desired working directory (e.g. c:\dynaform training), and enter a name for the database (e.g. training case.df). Press **Open** and the Dynaform database will be created.

Analysis Setup

1. After creating a new Dynaform database, the **Analysis Setup** dialog box is displayed.

2. In this box, select the default units, **MM, TON, SEC, N** and select **Inverted Draw** as the **Draw Type**. For this case, the remaining fields will be kept at their default settings. Press **Ok** when finished.

*Note: The **Draw Type** should correspond to the type of machine used to produce the actual work piece. This parameter defines the default moving direction of the punch and binders. If you are not sure, or are performing a new process, you should select **User Defined**. Refer to Section 2.2 for a description of draw type.*

II. Reading Geometry Data into a Database

Dynaform has the ability to translate the following types of input files:

1. Dynaform Database (*.df)
2. FEMB Database (*.fmb)
3. Iges (*.igs, *.iges)
4. VDA (*.vda)
5. DYNAFORM/FEMB Line (*.lin)
6. LS-DYNA (*.dyn, *.mod)
7. DYNAIN File (*.din)
8. NASTRAN (*.dat)
9. LS-Nike 3D (*.nik, *.mod)
10. C-Mold (*.fem)
11. Moldform (*.mfl)
12. Ideas Universal (*.unv)

For this training case, we will be reading in Dynaform/FEMB Line data. If you have not uncompressed the training cases provided by ETA, or are not sure if you have, follow the next step, otherwise proceed to Step 2.

STEP 1 – Uncompressing the Training Files

1. Open the folder C:\Program Files\Dynaform\Examples (default directory).
2. You will see 11 files with the extension .exe. These are self-extracting files for use with this Training Manual, the Applications Manual and the PostGL Tutorial.
3. Double click on the TRAINING.EXE file. The example files for the exercises in this manual will automatically be extracted and placed in the file TRAINING.

STEP 2 – Reading the Data in

1. From the menu bar, select **File➜Open.**
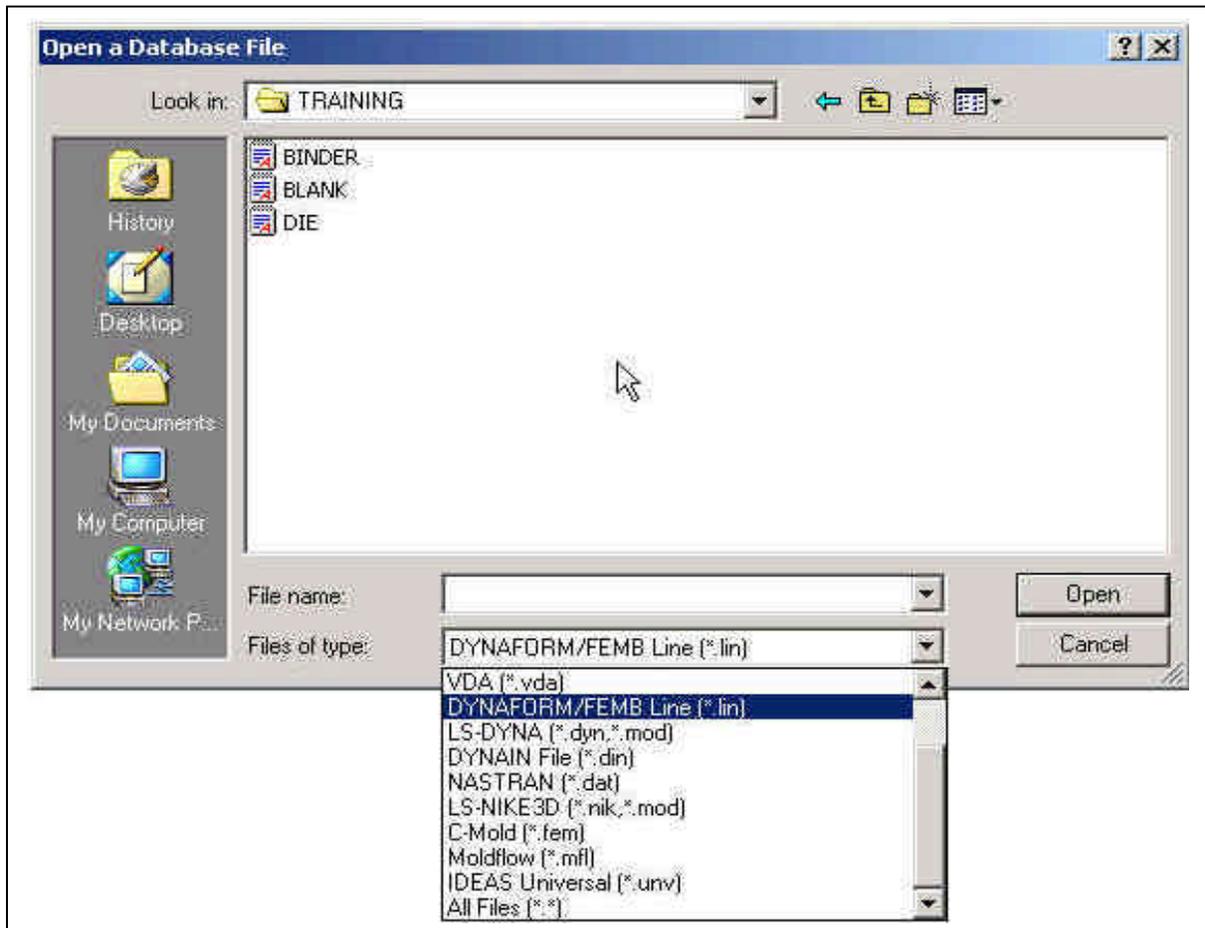
2. Move to the directory **C:\Program Files\Dynaform\Examples\TRAINING** (default directory) and select **DYNAFORM/FEMB Line (*.lin)** from the **Files of Type** menu. There should be three files displayed.
3. From the list of files, select **BLANK.LIN** and then press **Open.**

4. If asked, **"DO YOU WANT TO APPEND THE CURRENT DATABASE WITH THE INPUT FILE?"** Click **Yes.** The geometry will be displayed.

5. Repeat this step for each of the remaining files. Be sure to append each one to the current database.

6. Now that you have read in all of the line files, switch to the isometric view by selecting the **Isometric** icon on the **Toolbar**. Verify that your display looks the same as the picture below.

Note: *The colors may be different. Other functions on the **Toolbar** will be discussed further in the next section. You can also refer to the **Dynaform User's Manual** for information on all of the **Toolbar** functions.*
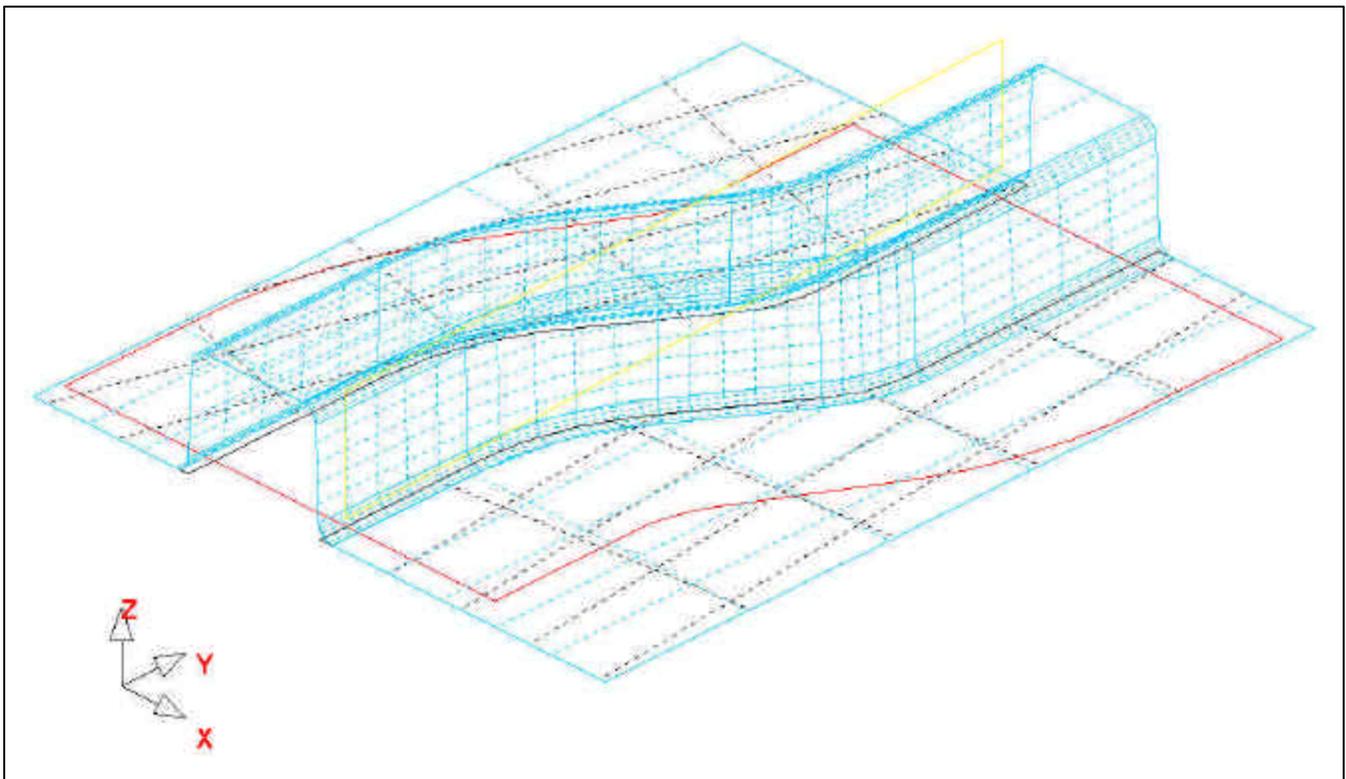
III.   Practice Some Auxiliary Menu Operations

Now that you have read in the files you will need to start pre-processing this simulation, there are some basic functions and menus you should familiarize yourself with.

Save/Save as…
It is a good habit to save often. To save in Dynaform, select **File➜Save** or **File➜Save as…**

```
New...
Open...
Close

Run LS-DYNA From Input File

Save
Save As...
Copy to Clipboard

Print Setup...
Print...

1 C:\Iges Files\From Dell 13\Retrim.df
2 C:\Iges Files\From Linux 4\Box Side.df
3 C:\Program Files\...\CASE2\case2.df
4 C:\Program Files\...\CASE2\case2abtg.df
5 C:\WINDOWS\...\Traincase\train.df
6 C:\WINDOWS\...\Traincase\training.df

Exit
```
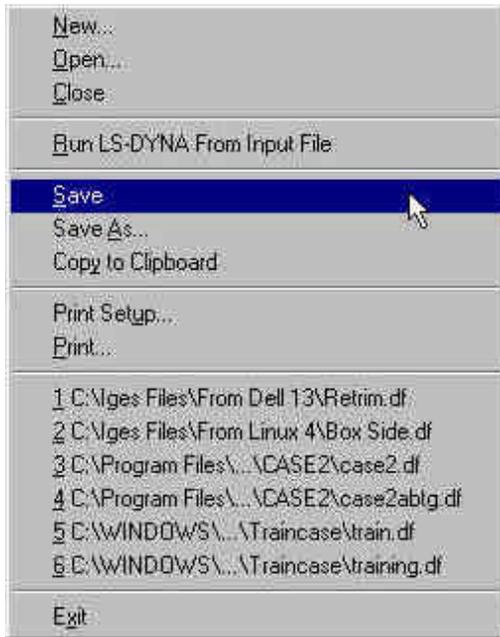
- Use **File➜Save** to save changes to an existing database file. The save command saves your file under the same name with which it was last saved and replaces the previous version.

- Use **File➜Save as…** to save to a new file under a new name.

Since you have already given your file a name, use the **File➜Save** command, and save the database.

View Manipulation

The view manipulation area of the **Toolbar** will quickly become one of your most visited spots in Dynaform. These functions allow you to change the display area's orientation. Hover over each icon to learn what it does. Also, take notice of the **Part Control** window (shown below) at the bottom of the screen. This is another area where you can manipulate the display area.


The following steps will help you become more familiar with the functions found on the **Toolbar**, and in the **Part Control** window.

1. Select **Isomeric** from the **Toolbar**. This places the displayed geometry in an isometric view, as shown earlier.
2. Rotate the geometry dynamically about the z-axis approximately 90° by using the **Rotate about Z-Axis** function. The result is shown below:
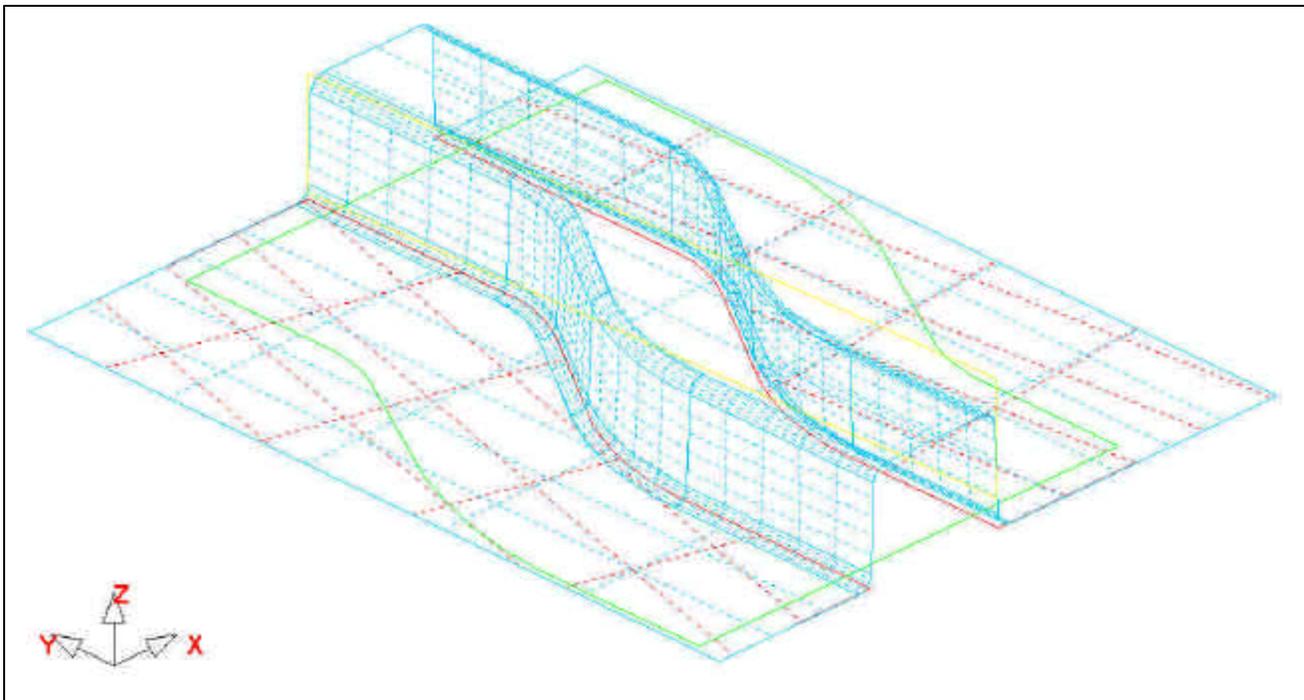
3. Select **View YZ Plane.** The result is shown below:
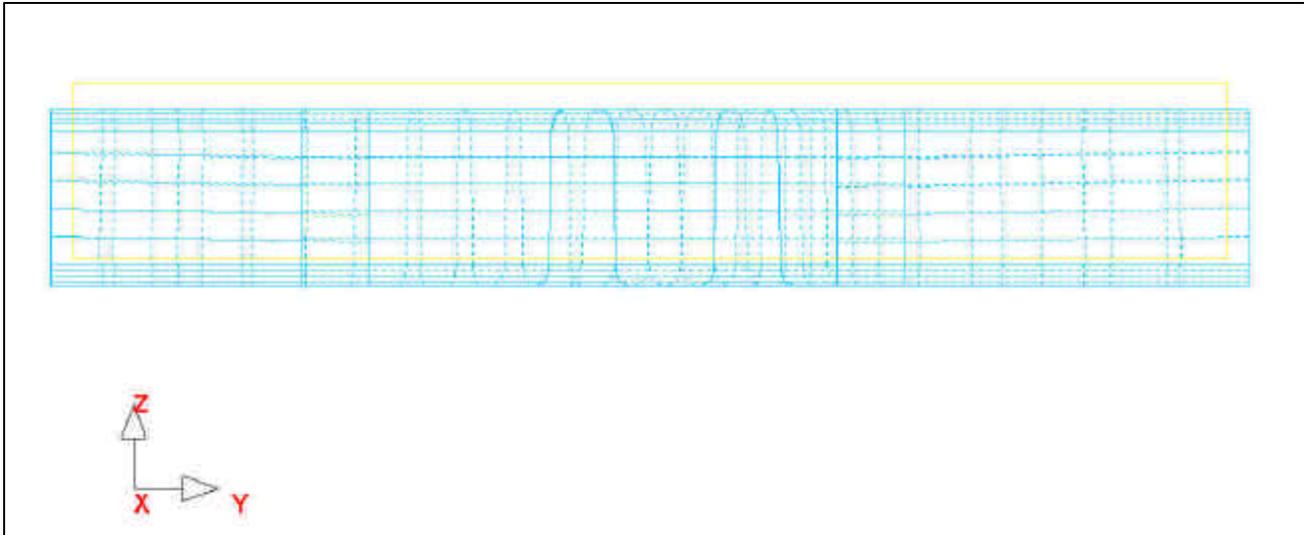
4. Select **Fill** from the **Toolbar**, this makes the displayed geometry fill the screen.

If you wish to, experiment with the other viewing options before moving on.

Turning Parts On/Off

All geometry in Dynaform-PC is based on parts. Every entity, by default, will be created or read into a part. The user should practice using the **On/Off** function, located in the **Part Control** window at the bottom of the screen.

1. From the **Part Control** window, select **On/Off**.

2. Notice the new **Option Bar** that appears just above the **Part Control** window.

This is the standard type of prompt Dynaform uses. Almost every function within the software will bring up a similar bar. Hover over the different icons to learn the name of each function. In the case of turning parts on and off, this bar gives you different ways of selecting what parts to turn on, and what parts to turn off.

3. Since the part **BLANK.LIN** has been generated using lines, you will have to use either the **Select by Line** or the **Select by Name** function.

4. First, use the **Select by Line** option to turn off the part, **BLANK.LIN**. Click on the **Select by Line** icon, then select a line in the part, **BLANK.LIN**. This part will be turned off.

5. Next, click the **Select by Name** icon, and then select **BLANK.LIN** from the list.


In the **Part On/Off** box, the parts that are turned on, are displayed in their color, and the parts that are turned off, are displayed in white. Press **OK** once you have selected **BLANK.LIN**.

6. Before we continue, verify that all available parts are turned on. Select the **Turn All Parts On** icon from the option bar.

7. After you have turned all the parts on, click the **End Select** icon on the option bar. This will end the current operation.
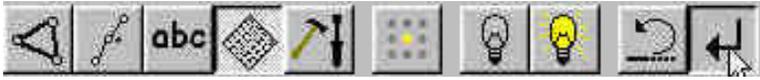
Editing Parts in the Database

The **Modify** command, located in the **Parts Control** window is used to define and edit part properties.

1. From the **Part Control** window, click on the **Modify** button.

The **Edit Part** dialog box will be displayed, with a list of all the parts that are defined in the database. The parts are listed by part name and identification number. From here, you can modify the part name, ID number, part color, set as the current part, or turn the part On/Off. You can also delete entire parts from this box.
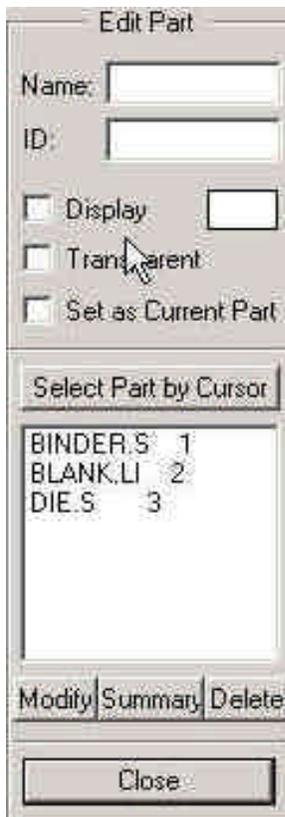
2. Select the part, **BINDER.S** from the part list. Change the color by clicking on the color box as shown below. Select a different color.
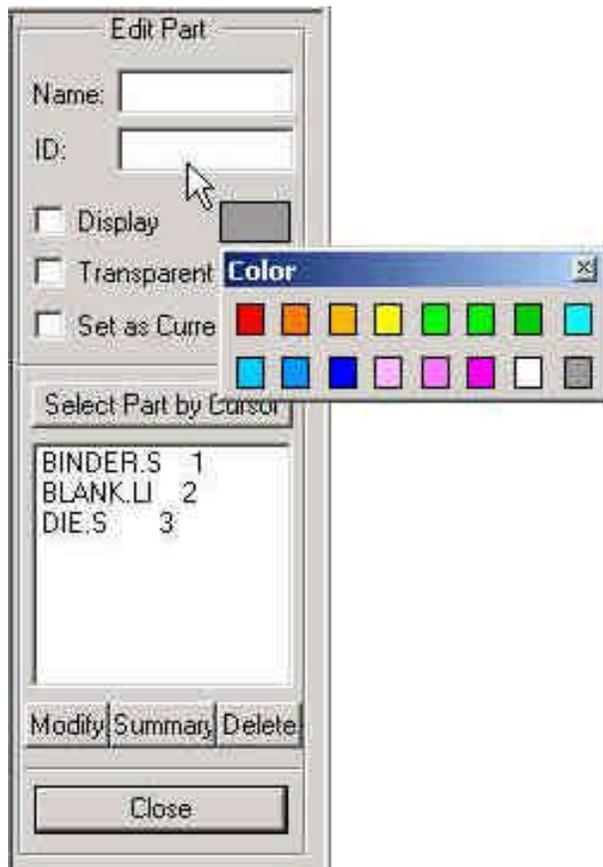
3. Once you have selected a different color, click on the **Modify** button located at the bottom of the **Edit Part** box. This button must be pressed after a change is made to a part. If the button is not pressed, the changes will not take affect.

4. Click **Close** to end this operation.

5. Save your database.

Current Part

All lines, surfaces, and elements that you create will automatically be placed into the current part. When creating new lines, surfaces, or elements, always make sure the desired part is set as current.

Note: *When auto meshing surfaces, the user does have the option of assigning the created mesh to the parts that contain the individual surface data. In other words, you can keep the mesh in the original parts, rather than have them all created in the current part. This will be dealt with later.*

1. To change the current part, click on the **Current Part** box in the **Part Control** window.

2. An **Option Bar** will be displayed.
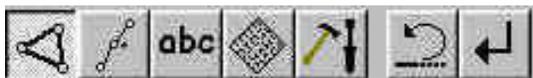
3. Similar to the **On/Off** option bar, this option bar allows you to select the current part in different ways. Hover over each icon to identify its function.

4. Set the part **BLANK.LI** as current by selecting the **abc** icon from the **Option Bar**.
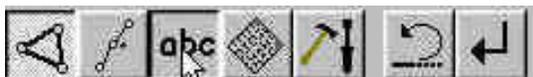
5. Now, select the part, **BLANK.LI** from the **Select a Part** box that is displayed. Press **OK** and the current part will be set.

6. Practice setting the current part as much as you would like. When you are done, turn off all of the parts except **BLANK.LI,** and set it as current.

IV. Meshing

Meshing from surface or line data is a very important step in a successful simulation. There are many methods of creating mesh that you will eventually become familiar with, but for now, we will be using the 4-Line and Surface Mesh functions.

Blank Meshing

Blank meshing is the most important part of meshing since the quality of results depends heavily upon the quality of the blank mesh.

1. Select the **Pre-Process** tab, then select **Elements** from the pull down list.

2. Select the **4-Line Mesh** icon from the **Pre-Process/Elements** menu.

3. When you click the **4-Line Mesh** button, a new **Option Bar** will be displayed, prompting you to select the desired lines.
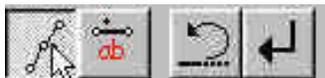

4. Make sure the **Line** option is highlighted (as in the picture above) and select the 4 lines that make up the blank in either a clockwise or counterclockwise order.

5. A dialog box prompts you to **"ENTER THE NO. OF DIVISIONS ALONG EACH LINE: L1, L2, L3, and L4"**
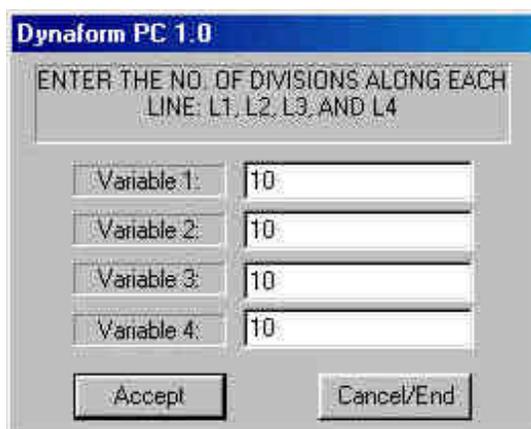
6. The default setting is 10 divisions along each side. This would create a perfect 10x10 mesh that contains 100 elements. Using the default in this case will not be small enough. Use instead, 15 divisions on the long sides, and 10 on the short sides. The variables 1, 2, 3 and 4 refer to the number of the line shown in the display area.

7. After you have entered the variables, press **Accept** and the mesh will be created. Dynaform will prompt once more; **"DO YOU ACCEPT THE GENERATED MESH?"** Verify that you have placed 15 divisions along the long sides and 10 along the short. Press **YES** if you have.
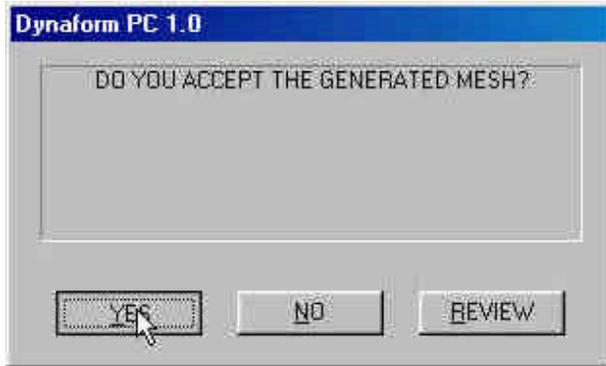
If you have accidentally entered the wrong variables, press **NO** when asked if you accept the mesh. Press **YES** when asked, **"WOULD YOU LIKE TO REMESH THE REGION?"** Correct the mistake and accept the mesh.
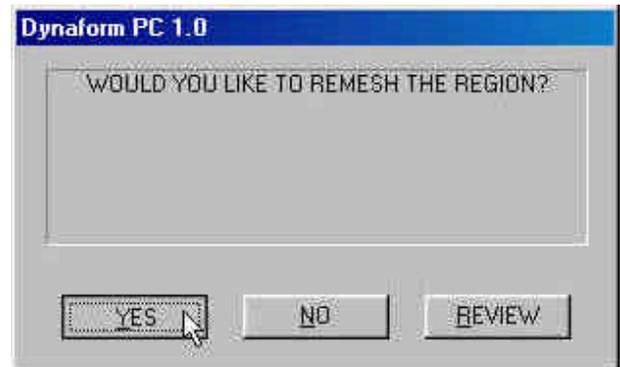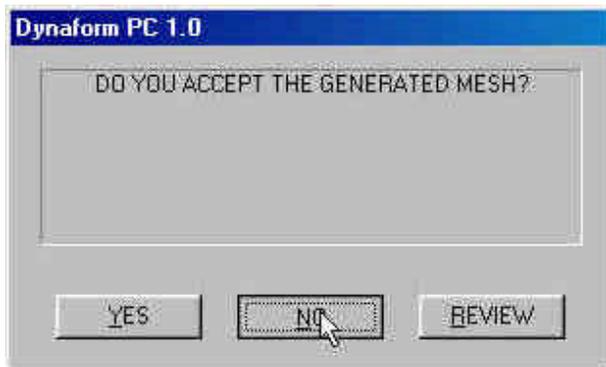
8. Check your mesh with the picture below.
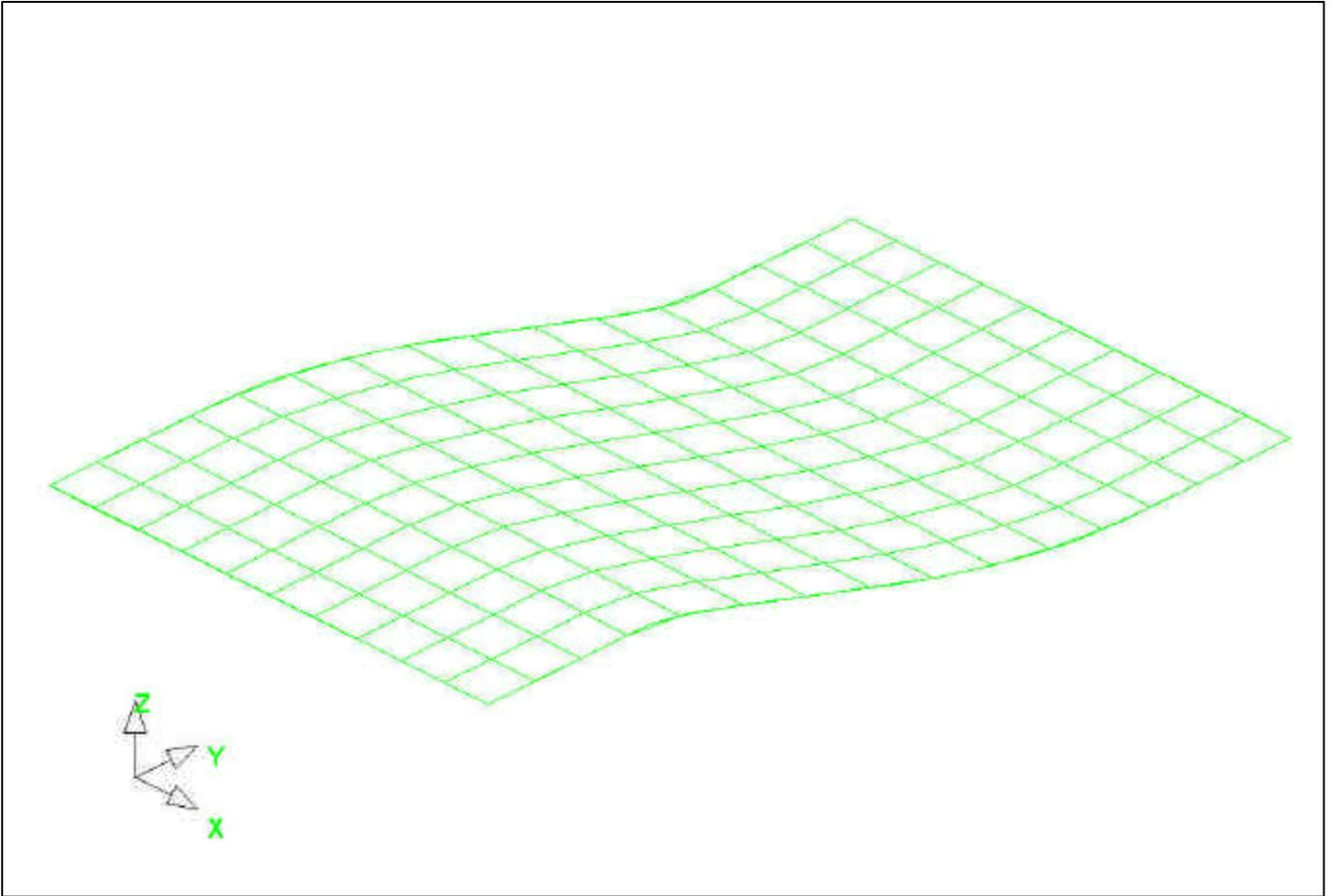
9. If your mesh looks the mesh above, press the **End Select** icon from the **Option Bar** to end the **4-Line Mesh** function.


10. Save the database.

Auto-Meshing Surface Data

Most of the meshing done in Dynaform is done using the **Surface Mesh** function. This function will automatically create a mesh based on surface data that you provide. This is a very quick and easy way of meshing the tools.

1. Turn off the part **BLANK.LI** and turn on the part **BINDER.S**. Set the part **BINDER.S** as current.

2. Select **Surface Mesh** from the **Pre-Process/Elements menu**.
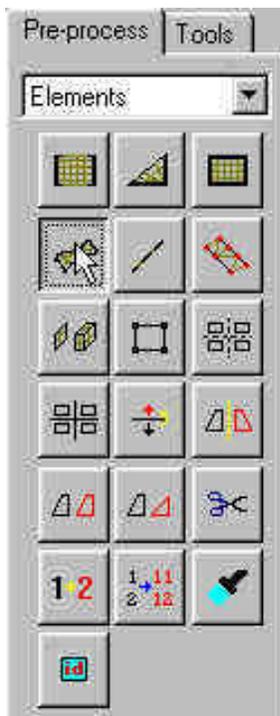
3. From the **Option Bar**, choose the **Select Displayed Surfaces** icon.

Notice all of the displayed surfaces will turn white. This verifies they have been selected.

4. In the **Surface Mesh** dialogue that is displayed, edit the **Max Size** field to be 16. The default values will be used for the remaining fields. Be sure that the **Mesh in Original Part** option is checked and click **Apply**.
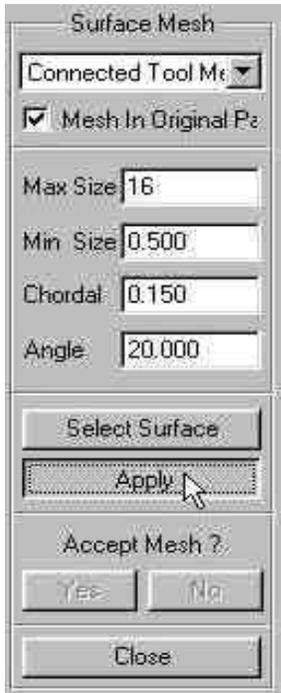
Note: *Chordal [deviation] controls the number of elements, and* **A**ngle *controls the feature line.*

5. The mesh will be created and will be displayed in white. To accept the mesh, click the **Yes** button where asked, **"Accept Mesh?"** in the **Surface Mesh** box.

Check your mesh with the mesh displayed below.
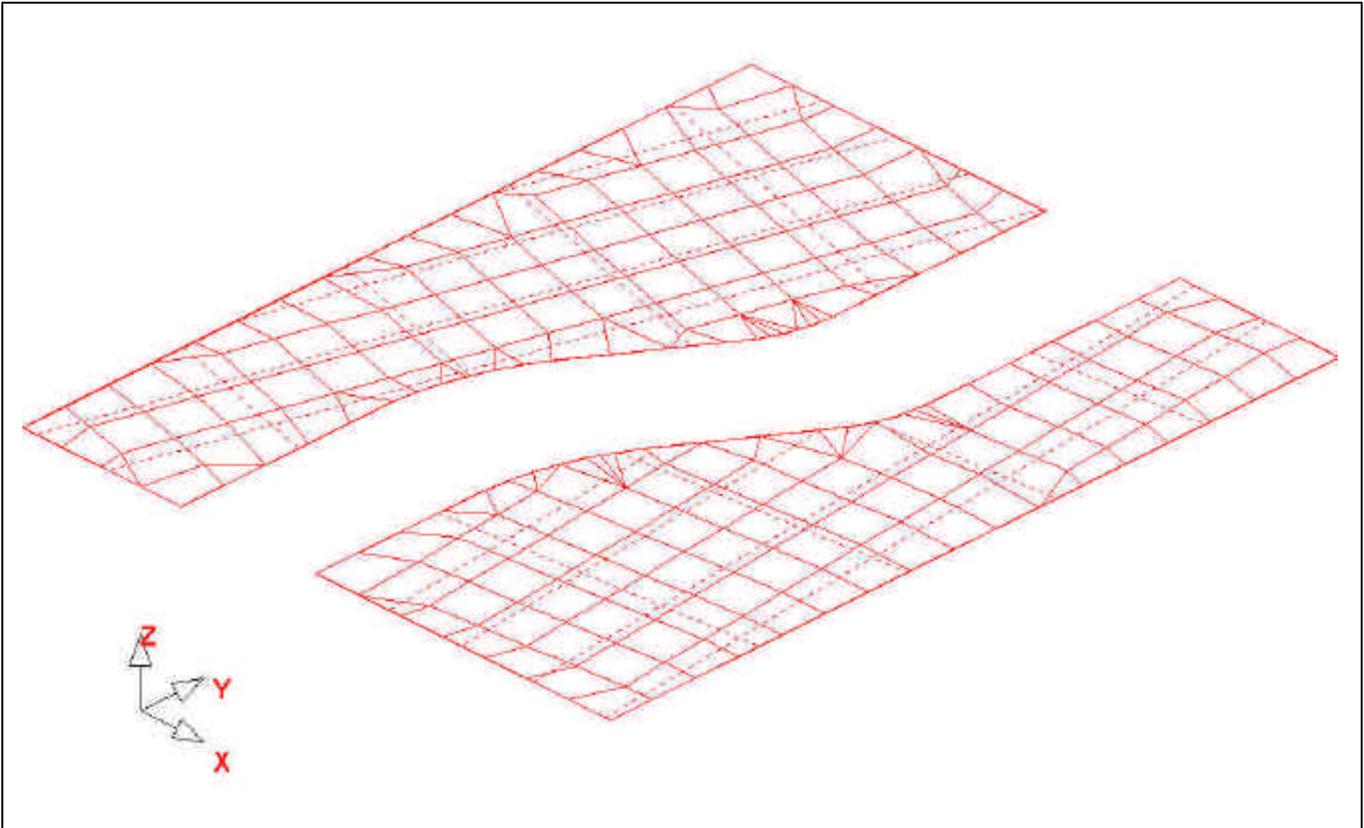
6. Press **Close** on the Surface Mesh box to end the function.

Auto-Meshing the Die Surface Data

1. Turn off all parts except for **DIE.S** and make it current.

2. Follow the previous 6 steps to auto-mesh the part **DIE.S** using 16 as the **Max Size**, 1 as the **Min Size**, and 0.005 as the **Chordal Deviation**.

3. Once you have generated the mesh with the given variables, close the **Surface Mesh** box and switch to the isometric view and verify that your mesh looks like the mesh shown below.
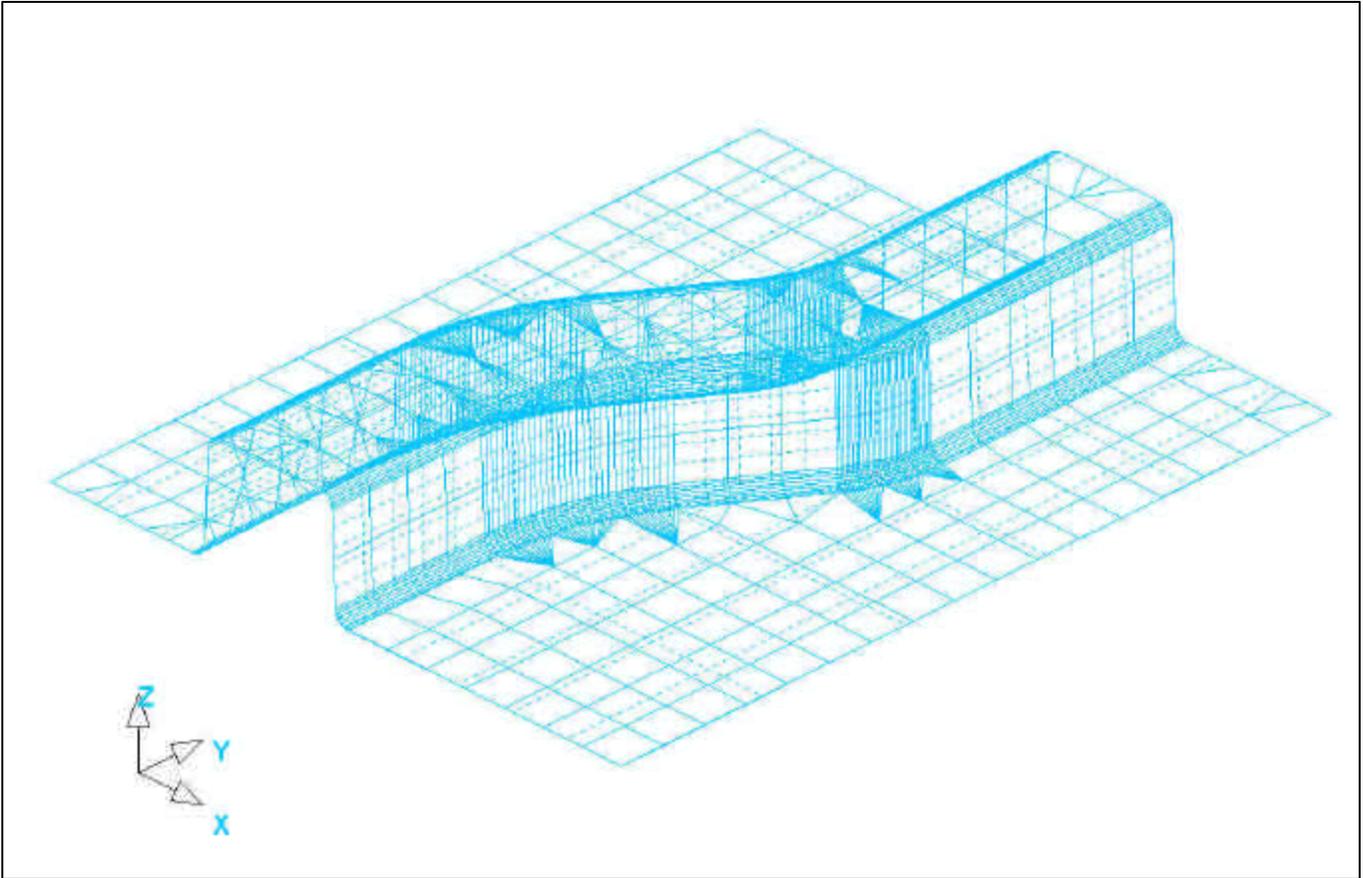
4. Now that we have all the parts meshed, you can turn off the surfaces and lines in the **Part Control** box. This makes things easier to view the mesh. Save the changes.

Click to deselect ────
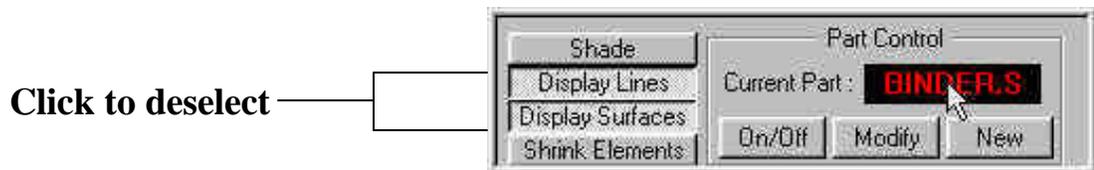
V.   Model Checking

Now that the mesh has been created, it needs to be checked in order to verify that there are not any defects that could cause problems later.

All the utilities you will be using to check the mesh are located under the check menu on the menu bar.
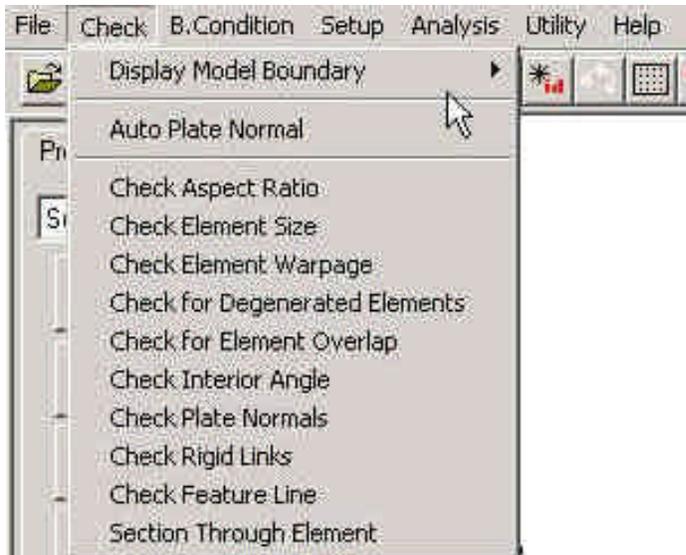
Auto Plate Normal

1. Select **Auto Plate Normal** from the **Check** menu. A new **Option Bar** will be displayed.

2. This prompt is asking you to pick an element to identify a part to be checked for consistency in the normal direction. Select an element on the part **DIE.S**. An arrow will displayed showing the normal direction of the selected element. A prompt will ask **"IS NORMAL DIRECTION ACCEPTABLE?"**
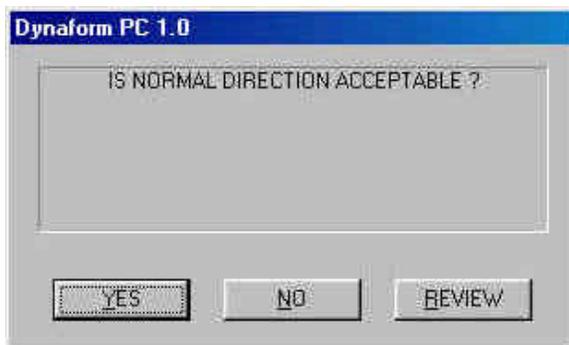
Pressing **YES** will check all elements in the part and reorient as needed to the direction that is displayed. Pressing **NO** will check all elements and reorient as needed to the opposite of the direction that is displayed. In other words, press **YES** if you want the normal to point in the direction of the displayed arrow, or **NO** if you want it to be the opposite. Since we will be using the normal offset function to create the punch, set the normal direction to the negative Z direction.

3. Now that the **DIE.S** elements are consistent you should check the rest of the parts in the database. Turn off all the parts and turn each one on by itself. Check the normal direction and make sure it is consistent.

Note: *For the remaining parts, the direction of the normal does not matter because we will not be using the normal offset option on them. You will only need to verify that the directions are consistent. So, just select **YES** when asked if you accept the direction.*

4. Once all the normal directions are consistent, turn on all of the parts and save the changes.

Display Model Boundary

This function will check the mesh for any gaps or holes, and highlight them so you can manually correct the problem.

1. Select **Check➜Display Model Boundary➜Multiple Surface**.
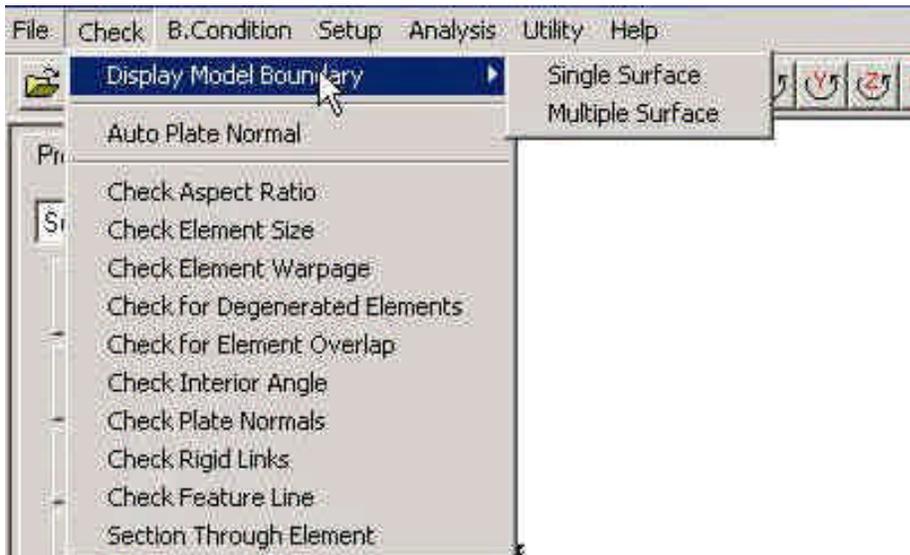
2. There should not be any gaps in any of the meshes. Select the isometric view and make sure that your display looks like the one shown below.
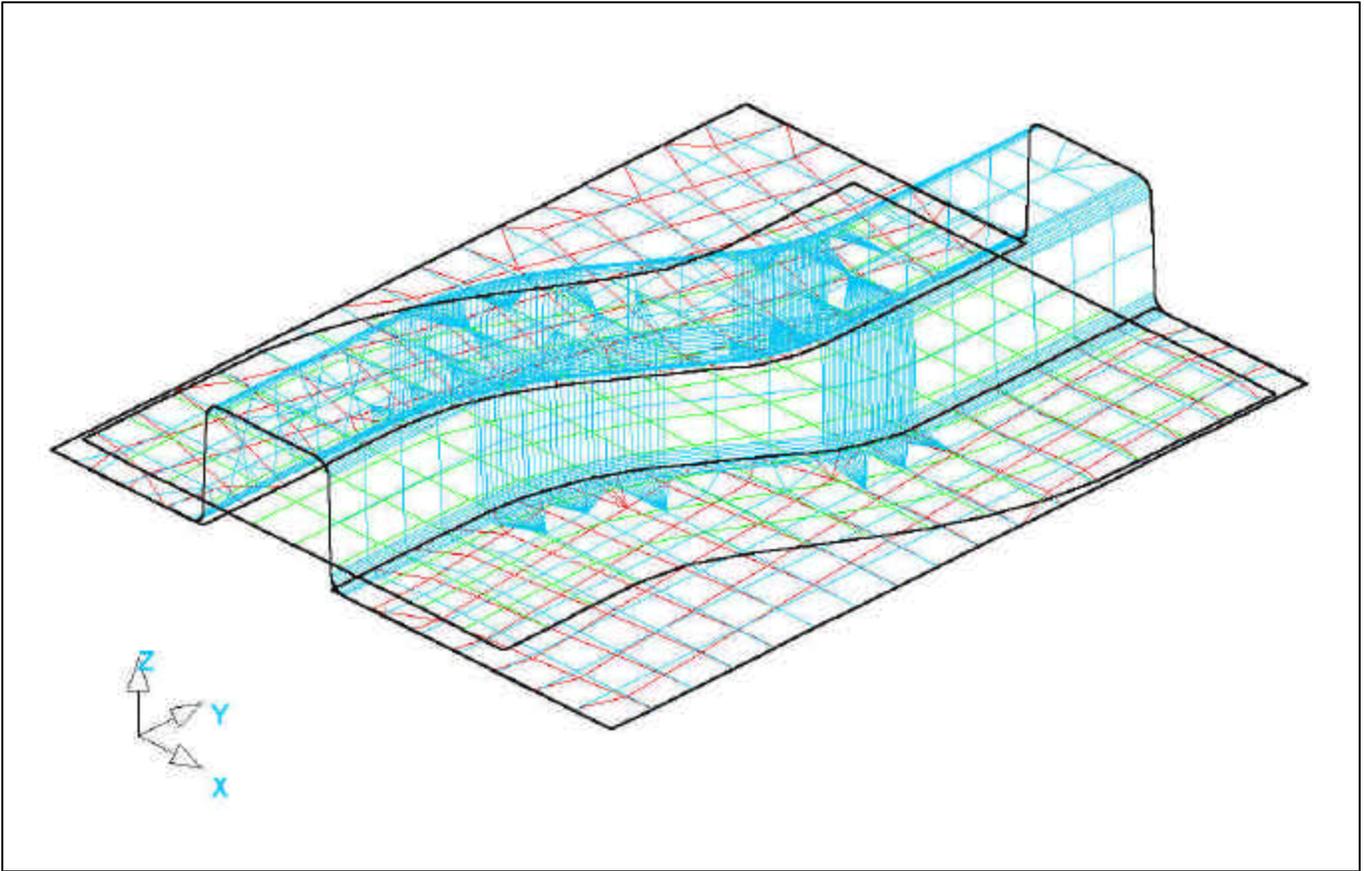
3. Now, turn off all of the parts and notice that the boundary lines are still displayed. This allows you to inspect for any small gaps that might be hard to see with all of the meshes showing. The results are shown below.
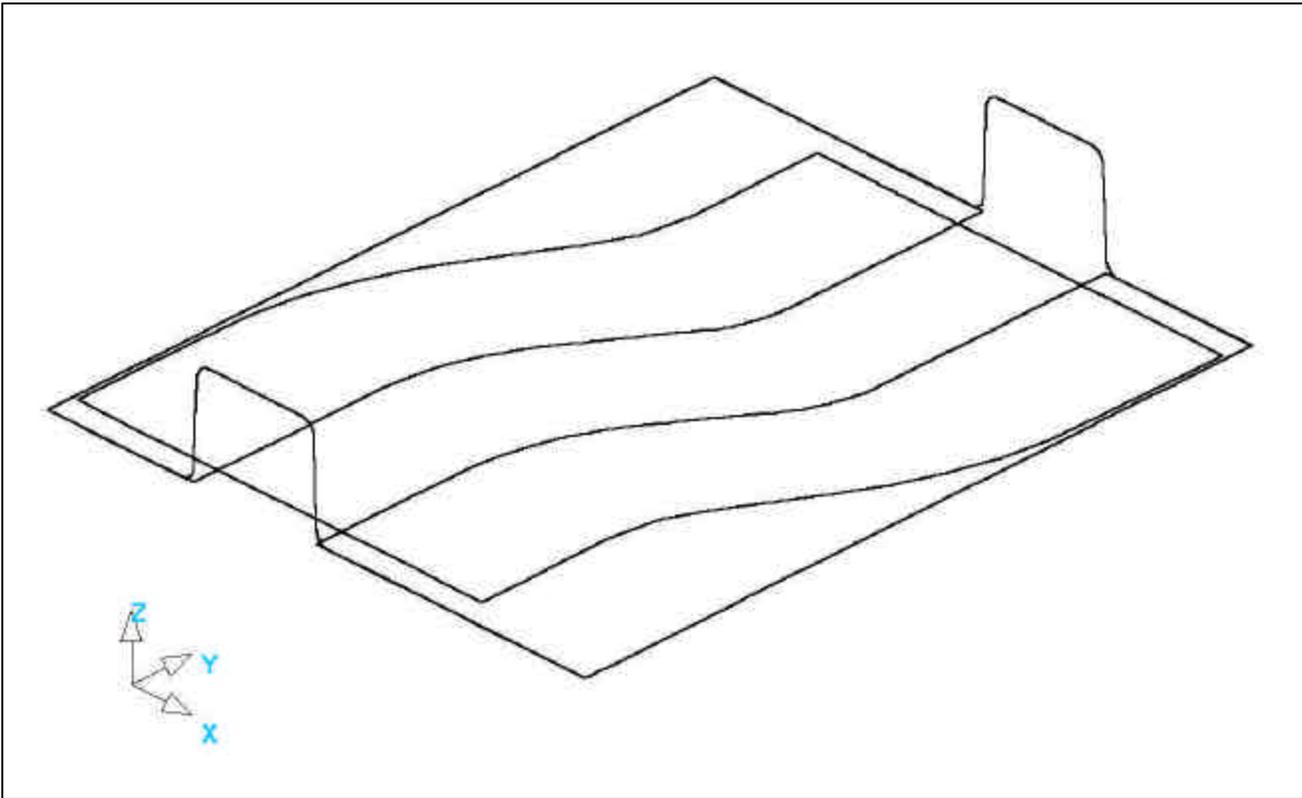
4. Turn only the part **DIE.S** back on and press the **Clear** button on the **Toolbar** to remove the boundary lines.

5. Save the database.

VI.   Offsetting the Die Mesh to Create the Punch

Now that the normal direction is set in the correct direction (negative Z), and we have checked for gaps in the mesh, we are ready to offset the Die mesh to create the Punch.

1. The first thing you need to do is create a new part called **PUNCH.** This part will hold the elements that we offset from the die. Click the **New** button in the **Part Control** window.

2. In the **New Part** box, enter "**PUNCH**" in the name field and be sure that the options, **Set as Current Part** and **Display**, are checked. Click **Create** and the part will be created.
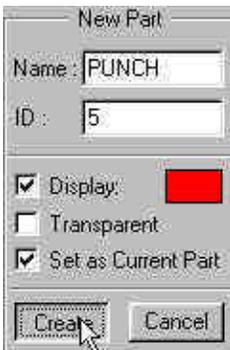
The part **PUNCH** has been created and we can now offset into this part.

3. Select the **Copy Elements** icon from the **Pre-Process\Elements** menu.

4. An option bar will be displayed, prompting you to select the elements that will be offset. You want to select all of the elements that make up the U-Channel of the part **DIE.S**. The easiest way to do this is to switch the view to the ZX plane on the **Tool Bar**, and then use the **Select by Drag Window** function on the **Option Bar**.
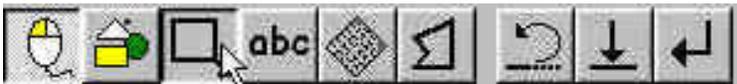
Refer to the picture below.
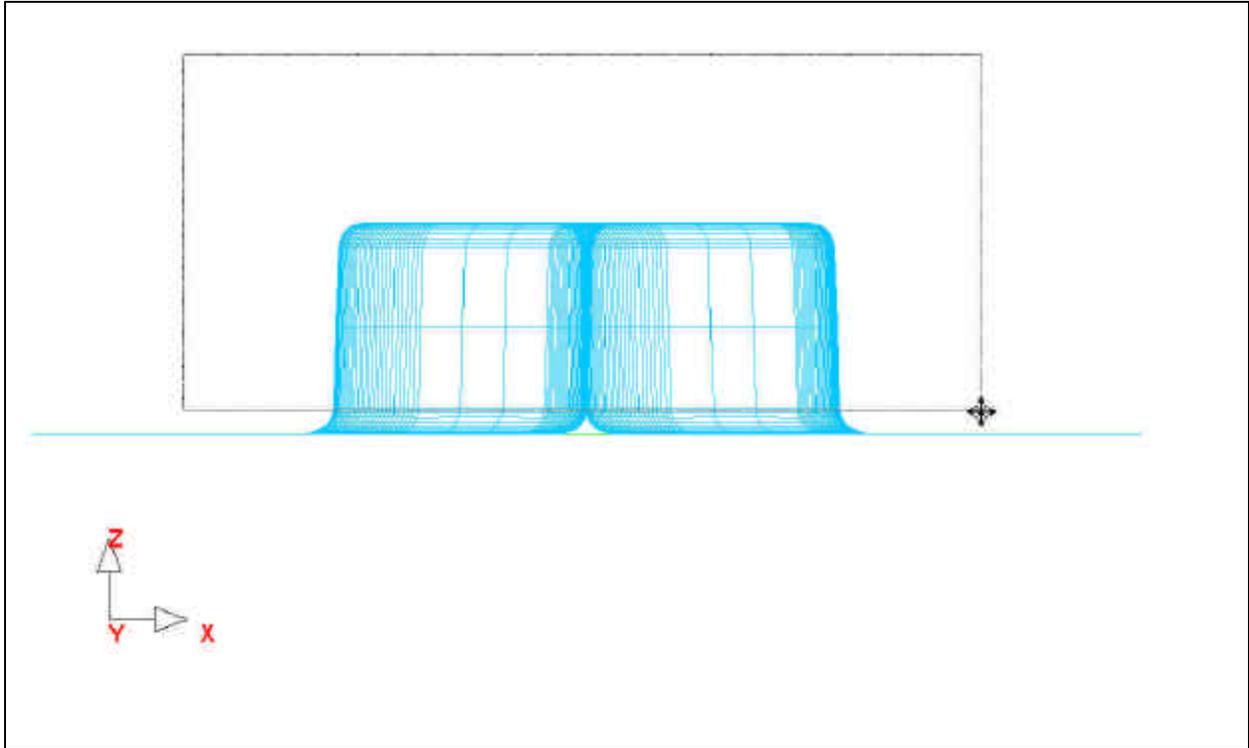
5. After you have drawn a drag window and selected the elements, Dynaform will ask you, **"DO YOU ACCEPT THE CURRENT DRAG WINDOW?"** Confirm that you have selected the correct region and press **YES**. If you have made a mistake, press **NO** and redraw the drag window.

6. After pressing **YES**, you will notice that the selected elements will turn white. Verify once more that these are the correct elements by checking your display with the display shown below.
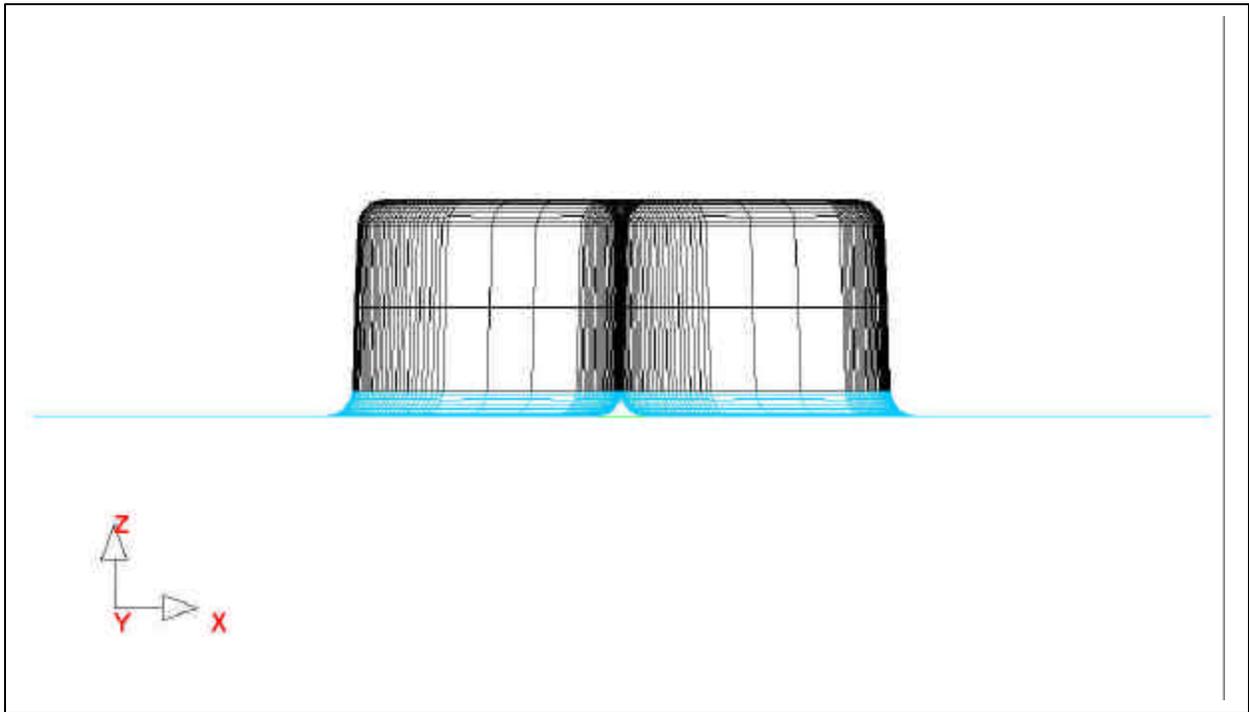
7. Press **End Select** after you have verified that the correct elements have been selected.
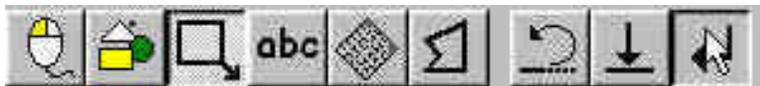
8. Enter **1** as the number of copies and press **Accept**.
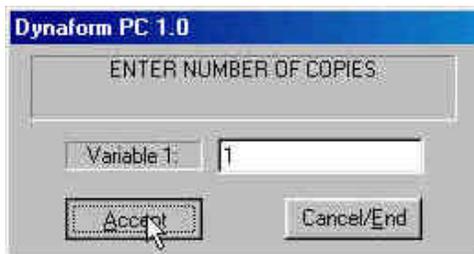
9. Select **Normal Offset** as the transformation option and press **Accept**.
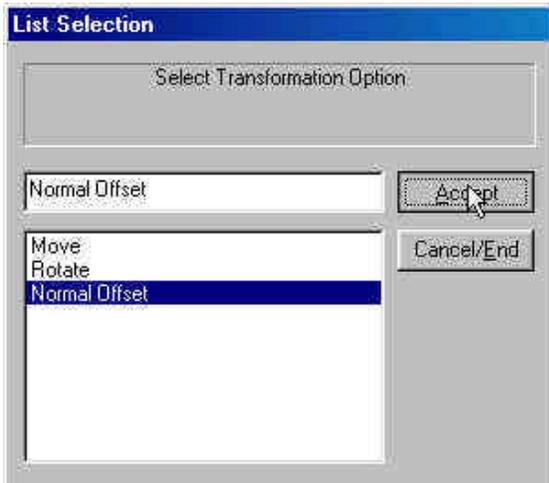
10. Enter the offset thickness. In Dynaform, we use the material thickness of the blank plus 10%. Since we will be using a blank thickness of 1, enter 1.1 in the box and press **Accept**.

Note: *We use 10% of the thickness for simulation because when we do the post-processing, sometimes wrinkle data is lost if there is not enough space between the punch and die after it has completed its travel path. If we use only the blank thickness as a gap, the punch will iron the blank, creating the impression that no wrinkling has occurred.*
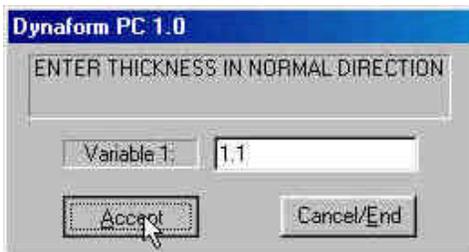
11. Dynaform will ask you, "**INCLUDE ELEMENT IN ITS ORIGINAL PART?**" Selecting **YES** will place all of the offset elements into the part **DIE.S,** pressing **NO** will place all of the elements in the current part. Make sure **PUNCH** is set as current, and press **NO**.
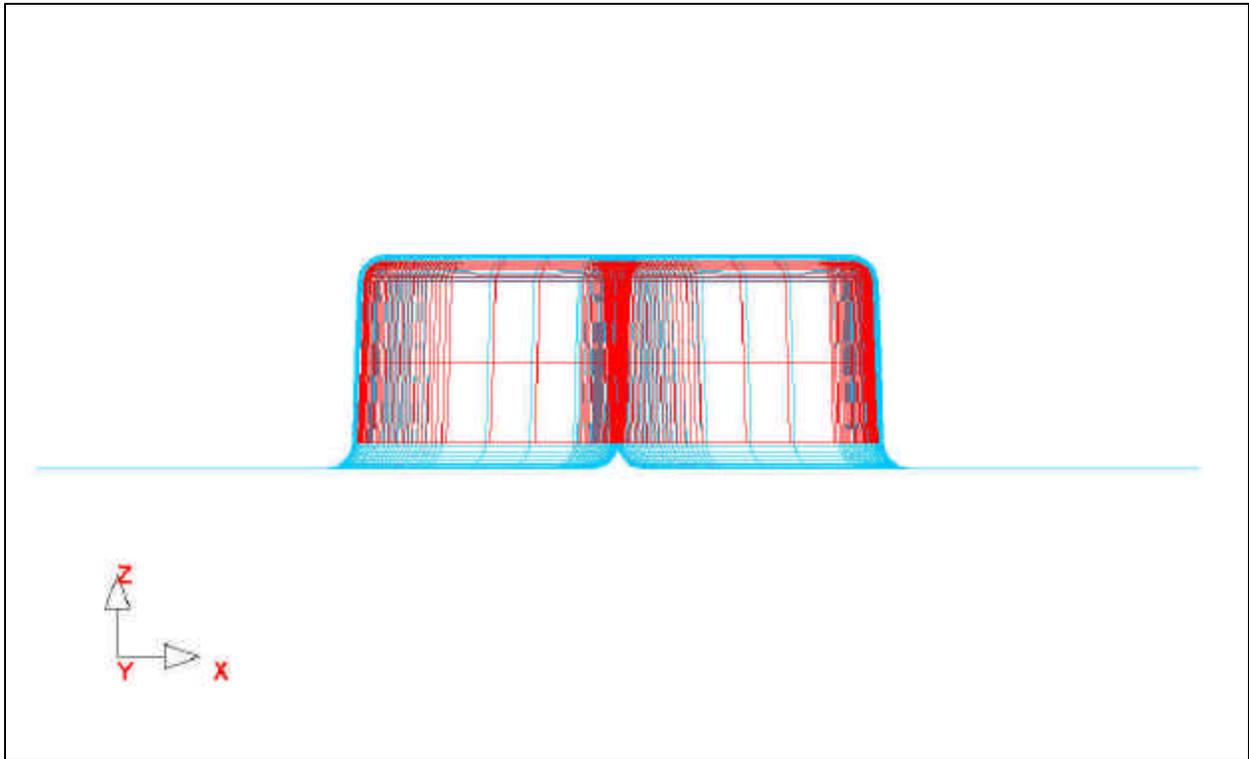
12. Check your display with the display below. If your results differ, **Right Click** in **the Display Area** and press **Undo Last**. Repeat the above steps to re-offset.

13. Turn off the part **DIE.S** so that only **PUNCH** is displayed. Switch to the isometric view. Check your display with the display below.
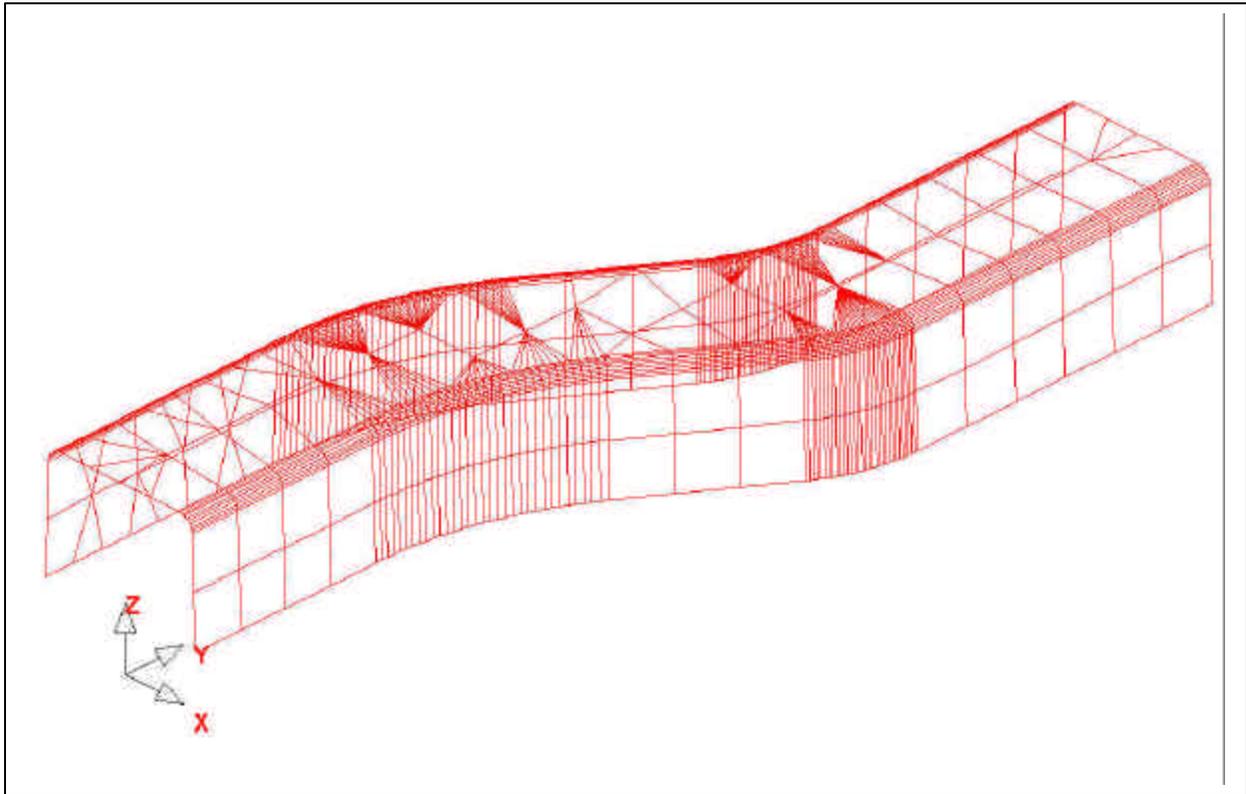
14. Save the changes.

V.   Tool Definition

Defining Parts as Tools

The parts BINDER.S, DIE.S, BLANK.LI, and PUNCH are all meshed and can now be defined as tools. After the parts have been defined as tools, we will setup the analysis parameters and start the solver.

1. Switch to the **Tools Menu** and select the **Define Tools** icon.

2. In the **Define Tools** box, select **Lower Ring** from the **Tool Name** list.

3. Press **Add**.

4. A new **Option Bar** will be displayed, prompting you to select which part will be defined as the Lower Ring. Choose the **Select by Name** (**abc**) icon.
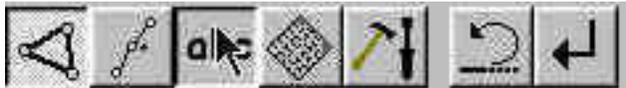
5. From the **Select Multiple Parts** box that is displayed, select **BINDER.S** and then press **OK**.
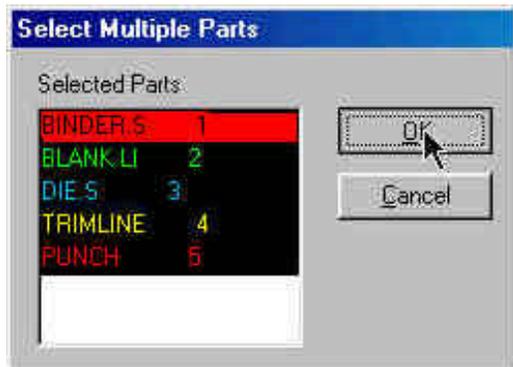
6. Press the **End Select** icon and the part **BINDER.S** will be defined as the Lower Ring.
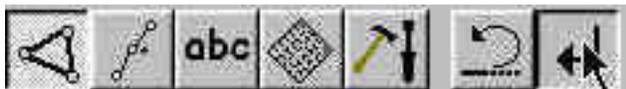
7. Repeat the above steps to define the Punch and Die. Remember to select the correct **Tool Name** in Step 2.

8. Once you have all of the tools defined, click **Close** on the **Define Tools** box to finish this step. Save the changes.

VII. Defining the Blank and Setting up Processing Parameters

Define the Blank

1. Select the **Define Blank** icon from the **Tools** menu.

2. Press **Add** in the **Define Blank** box.

3. A new **Option Bar** is displayed. Choose the **Select by Name** icon.
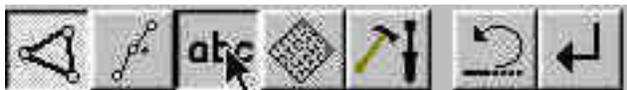
4. Select **BLANK.LI** from the **Select Multiple Parts** box and press **OK**.
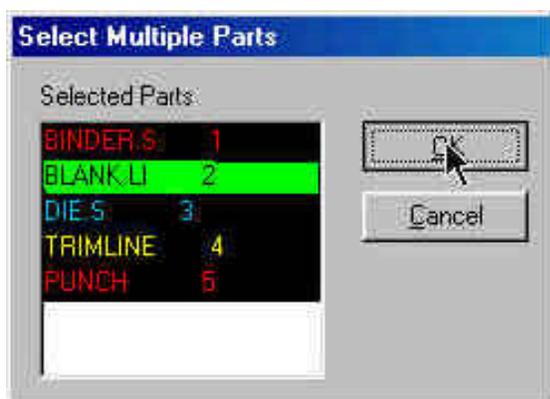
5. Press **End Select** on the **Option Bar** and the Blank will be defined.

Defining the Blank Material

1. The **Define Blank** box should still be opened, click on the button just below where it says **Material:** (This button will say **None** because the material has not been defined yet).

2. In the **Define Material** box, enter a name for the material, or use the default. From the **Material Type list**, be sure type **36** is selected and click **Add**.

3. The **Material Type 36 Definition Card** will be displayed. In this box, you can edit any values you want. To edit a field, left click on the field and change the value in the **Current Value** box. Change **MATERIAL PARAM P2 (N)** to **0.23**. Leave all other fields at their default values and press **Accept**.
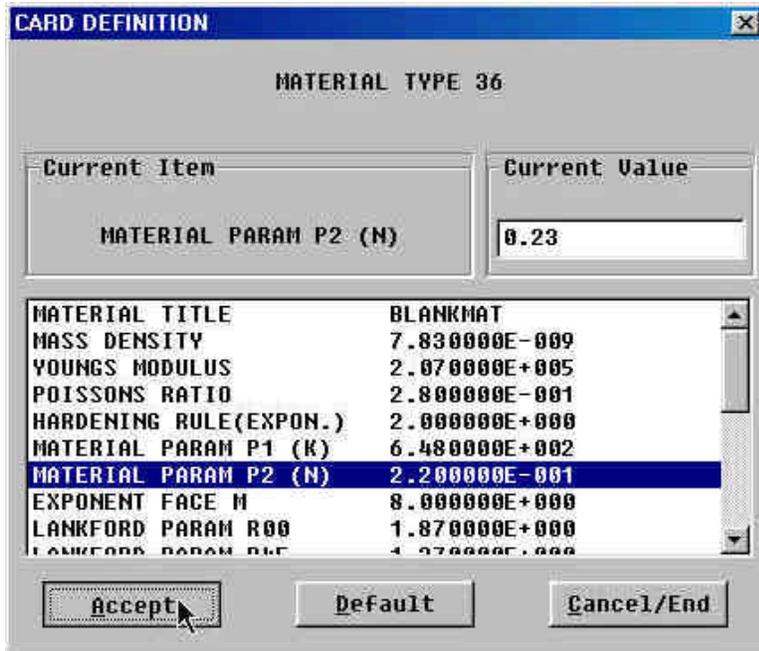
4. Press **Ok** in the **Define Material** box to complete material definition. Save the changes.

Define Blank Property

1. The **Define Blank** box should still be opened, click on the button just below where it says **Property:** (This button will say **None** because the property has not been defined yet).

2. In the **Define Property** box, enter a name for the material, or use the default. Be sure **Belytschko-Tsay** is selected as the formulation type, and click **Add**.

3. The **Belytschko-Tsay Definition Card** will be displayed. Use the default values for all of the listed parameters except for the **Uniform Thickness**. Edit the **Uniform Thickness** to be **1** and press **Accept**.
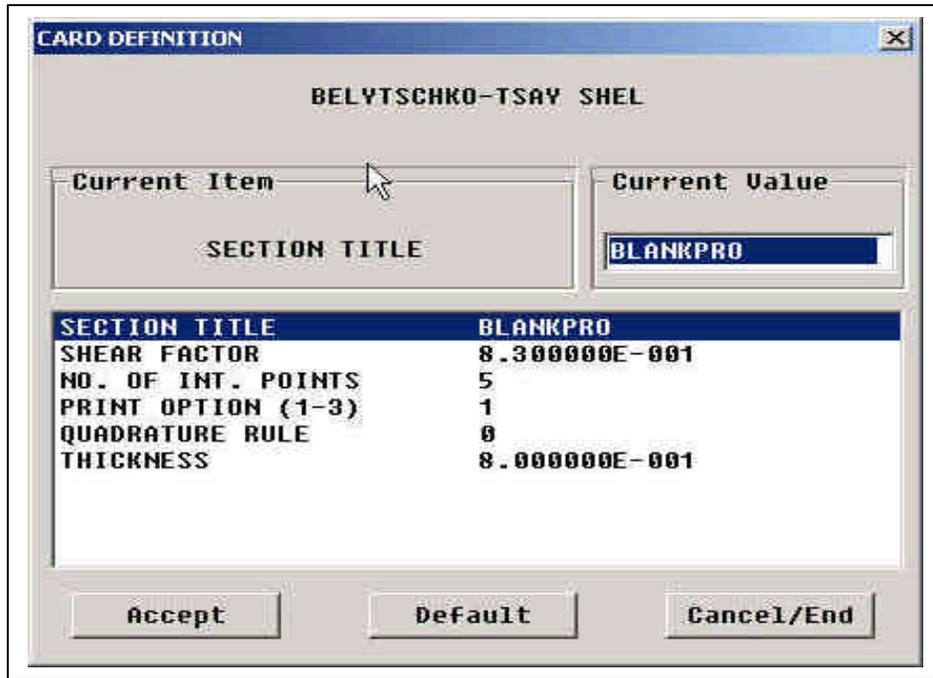
4. Press **Ok** in the **Define Property** box to complete property definition. Save the changes.

5. With both the blank material and blank property defined, you can now press **Close** in the
    Define Blank window to end blank definition.

Tools Summary

1. You can verify that you have defined all of the needed tools by selecting the **Tools
    Summary** function on the **Tools** menu.

2. Verify that all tools other than UPPER RING have been defined, then click **Close**. Save the changes.

Auto Positioning the Tools

Now that all of the tools have been defined, we will need to place them in their correct position.

1. Turn on all of the parts in the database and select the isometric view.

Click the **Auto Position** icon in the **Tools** menu.

40
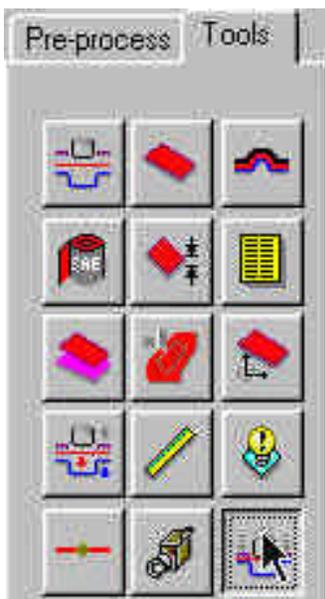
2. The **Auto Position Tools** box will be displayed. In this box, you will need to define which part will be the Master Tool and which parts will be the Slave Tools. The Master Tool is the tool that will not move while you Auto Position and should be the Blank. Select **BLANK** in the **Select Master Tool** box and then select the remaining tools in the **Select Slave Tools** box. Once you have selected the correct Master and Slave Tools, be sure that **Z** is selected as the moving direction and enter a **Gap** of **1.1** (The gap value should be greater than the Blank Thickness). Press **Apply** and the tools will be positions.

Check your display with the display below.
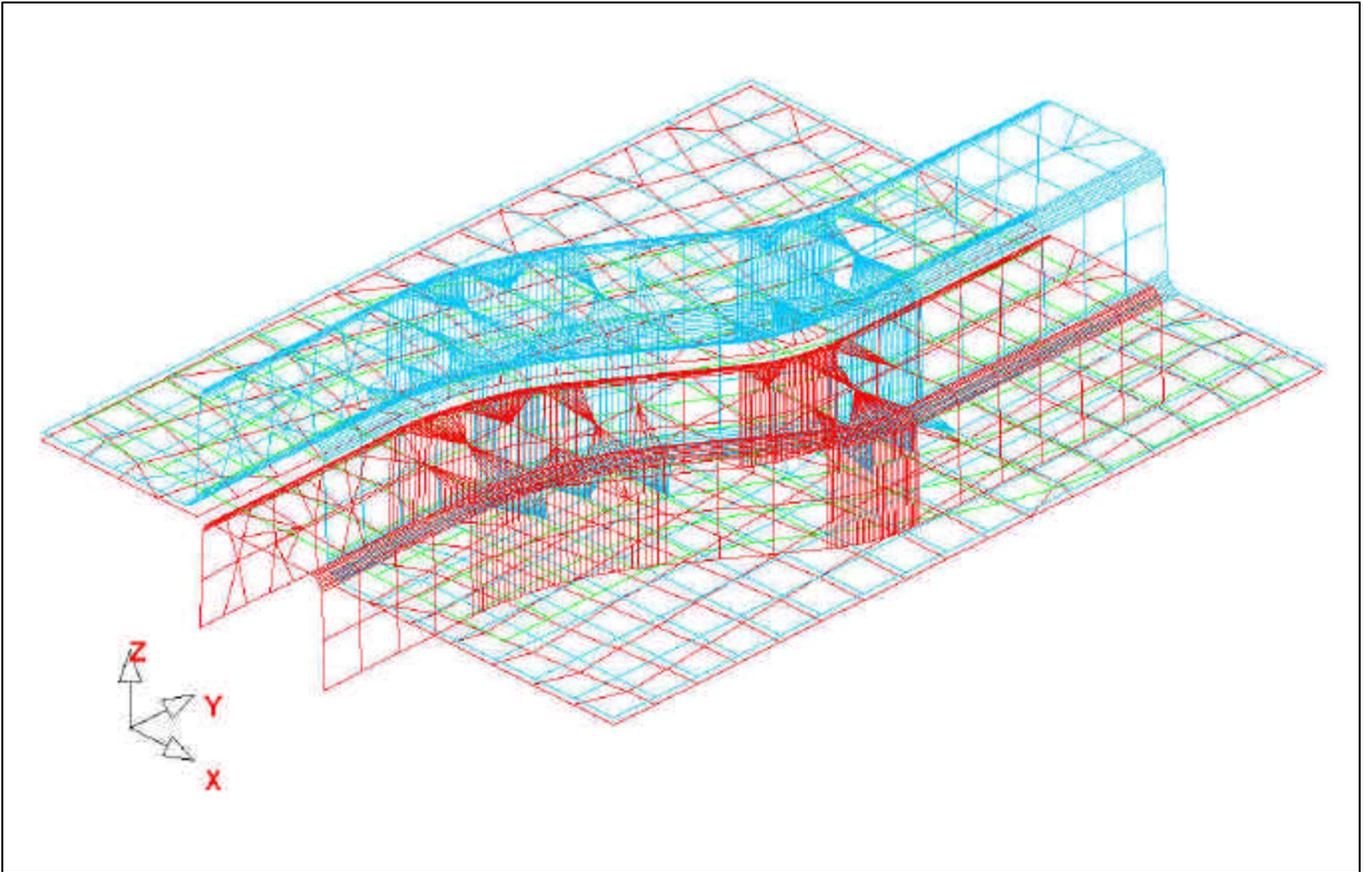
3. Press "Cancel" to exit the auto position menu.
4. Save the changes.

Measuring the Punch Moving Distance

Now that the meshes have been created and the parts have been defined as tools, we can setup the motion curves. The first step will be to find the moving distances for the parts.

1. Select the **Min. Distance** icon from the **Tools** menu.

2. The **Min. Distance** box will be displayed. Select **Z** as the direction to measure in, then highlight **Punch**, then **Die**. This will measure the distance, in the Z-direction, between the Punch and the Die, and display it in the **Distance** box.

3. The total distance between the Punch and the Die is approximately 42.2. To find the Punch travel distance, subtract the Blank thickness + 10% of the Blank thickness. After doing so, we get a Punch travel distance of approximately 41.1. Take note of the number you get.

Note: *We use 10% of the thickness for simulation because when we do the post-processing, sometimes wrinkle data is lost if there is not enough space between the punch and die after it has completed its travel path. If we use only the blank thickness as a gap, the punch will iron the blank, creating the impression that no wrinkling has occurred.*

4. You can now press **Close** in the **Min. Distance** box, and end this step.

Define Punch Velocity Curve

1. From the **Tools Menu**, select the **Define Tools** icon.

2. Select Punch in the **Tool Name** list and then select the **Define Motion Curve** icon.
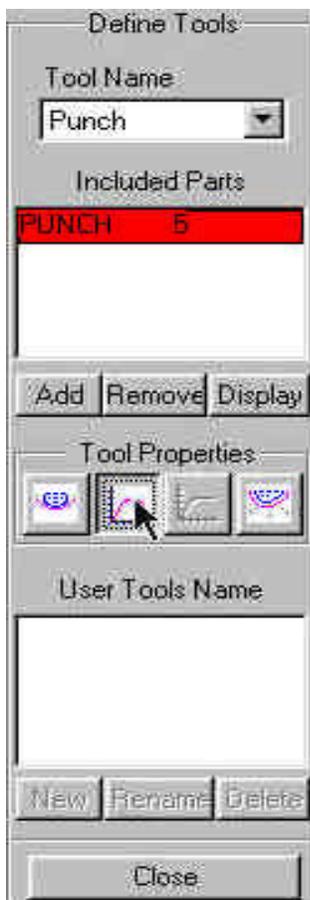
3. Select **Z** on the bottom of the **Tool Motion Curve** box to define the moving direction, then press **Auto**.

4. Use the default **Curve Type** (Tropezoidal) and keep the begin time of 0.000E+000. In the **Velocity** field, enter 8000 (this is in mm per second). For the **Stroke Dist.**, use the value that we found after measuring the Punch travel distance and subtracting the Blank thickness + 10%. The value should be approximately 41.1. Once you have entered the values, press **Yes** and a new Velocity vs. Distance motion curve will be created and displayed.
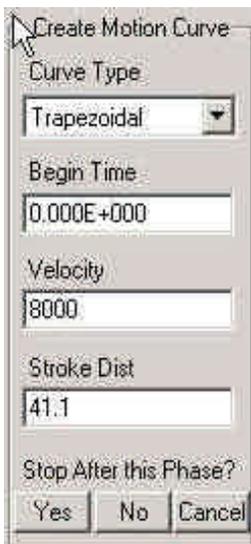
5. Verify that your graph is identical to the graph shown below.
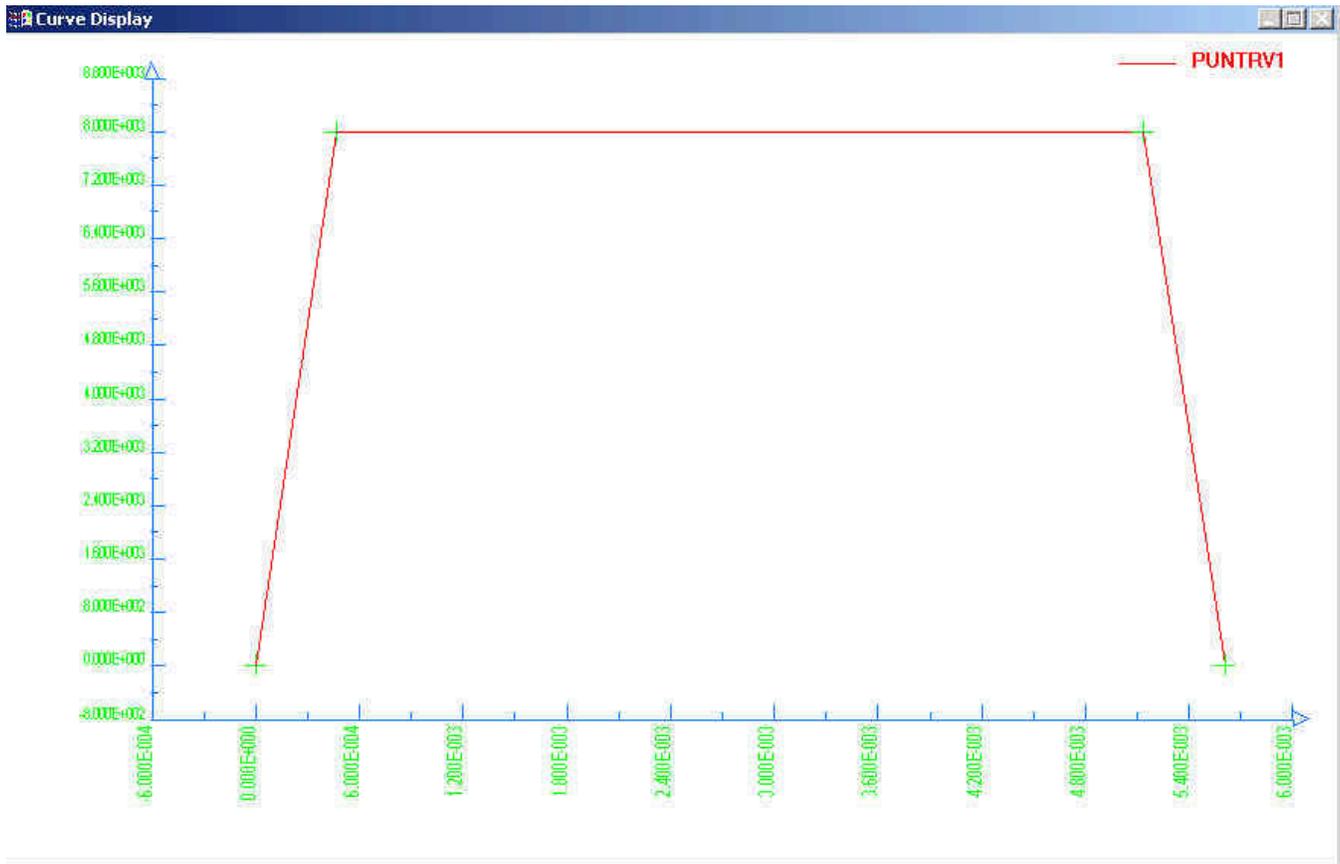
6. Press **Ok** in the **Curve Show Property** box to return to the **Tool Motion Curve** box. From here, press **Ok** once more to return to the **Define Tools** box. Do not close this box, the next step will start from here.

Defining Lower Ring (Binder) Force Curve

Now that the Punch travel curve has been created, we can create the force curve for the Lower Ring.

1. From the **Define Tools** box, select **Lower Ring** from the **Tool Name** menu and select the **Define Force Curve** icon.
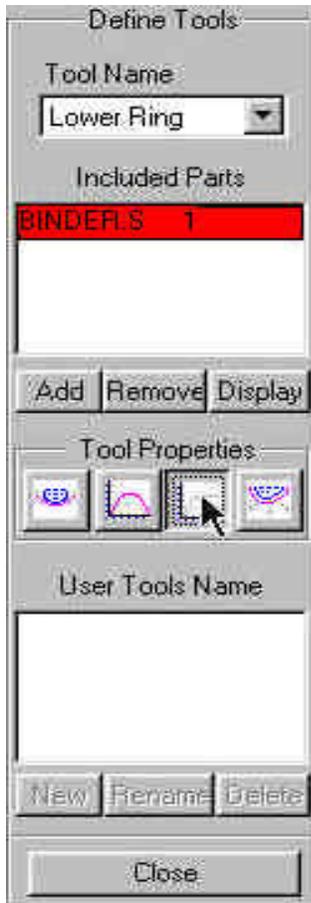
2. In the **Tool Force Curve** box, select **Z** to define the moving direction and then press **Auto**.

3. Enter 200000 (N) in the **FORCE** and press **Create**.
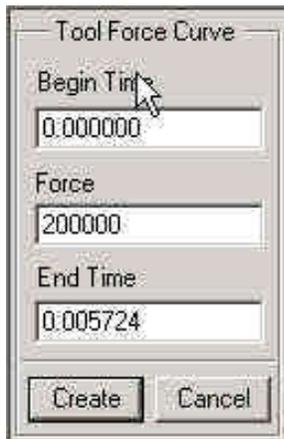
4. The Lower Ring force curve will be displayed. Verify that it is identical to the curve shown below.
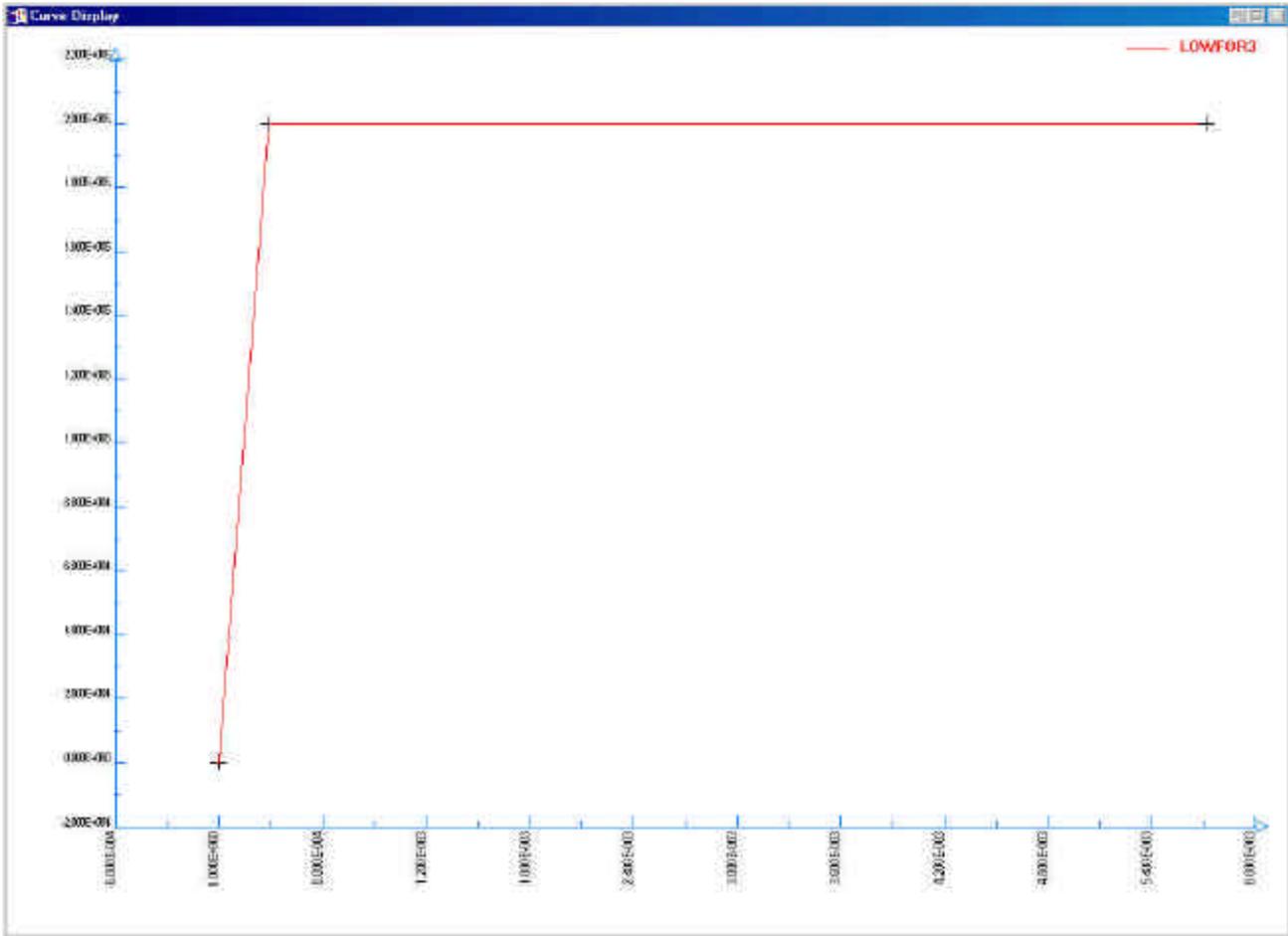
5. Press **Ok** in the **Curve Show Property** box to return to the **Tool Motion Curve** box. From here, press **Ok** once more to return to the **Define Tools** box. You can now press **Close** in the **Define Tools** box; we are done defining motion curves for the tools.

6. Select "OK" and "Close" back to the Tool menu.

7. Save the changes.

Preview Tool Animation

We have now completed all of the pre-processing outside of setting up the final simulation parameters and submitting the job. Before we can do that however, we should verify that the tools are moving correctly, based on the motion curve information we have provided.

1. Select the **Tools Animate** icon from the **Tools** menu.

2. You will be prompted to "**Please Enter Number of Frames (1~20)**". **Accept** the default value of 10 frames.

Note: *Depending on the speed of the machine you are running Dynaform on, the animation might move too quickly. If this is the case, enter a larger number of frames.*
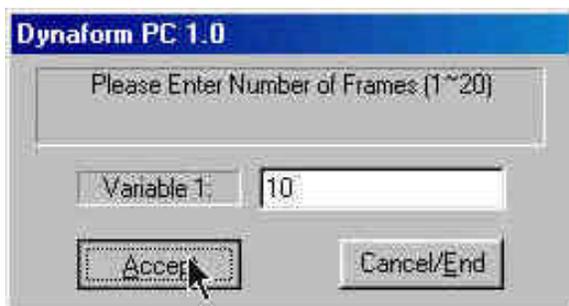
3. You can change the view while running the preview animation. Also, notice the frame controls on the **Option Bar**. Verify that the Punch is moving in the Z-direction, and make sure it is moving the full distance into the Die. Since we are using a force curve for the Binder, you will not see it move. When you are done viewing the animation, press the **End Select** button on the **Option Bar**.

4. Save the database.

VIII. Running the Analysis

Now that we have verified that the tool motion is correct, we can define the final parameters and run the analysis.

Running the Analysis with Adaptive Mesh

Adaptive mesh allows for more accurate results by re-meshing the model as needed. In other words, when the solver comes across an area on the Die that will demand a finer mesh to capture the geometry, it will split the original mesh to create finer, smaller elements.

1. Select **Analysis→Run LS-DYNA…** from the **Menu Bar**.

2. The **Analysis Parameters** box will be displayed. Turn on **Adaptive mesh** by clicking the check box.

3. Select the **Adaptive Parameters** button as shown below.
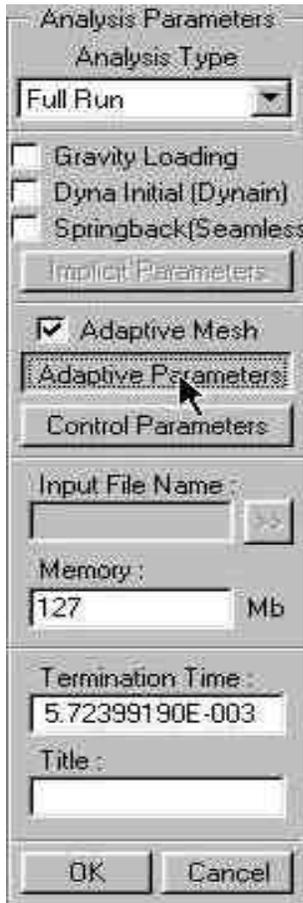
4. In the **ADAPTIVE CONTROL PARAM. CARD**, edit the **REFINEMENT LEVEL** to be 3. This means that the mesh will split up to 3 times if needed. The higher the **REFINEMENT LEVEL** the better the results but the longer the processing time. Since this is a simple part, 3 will be fine. The default values will be used for the remaining parameters. Press **Accept**.
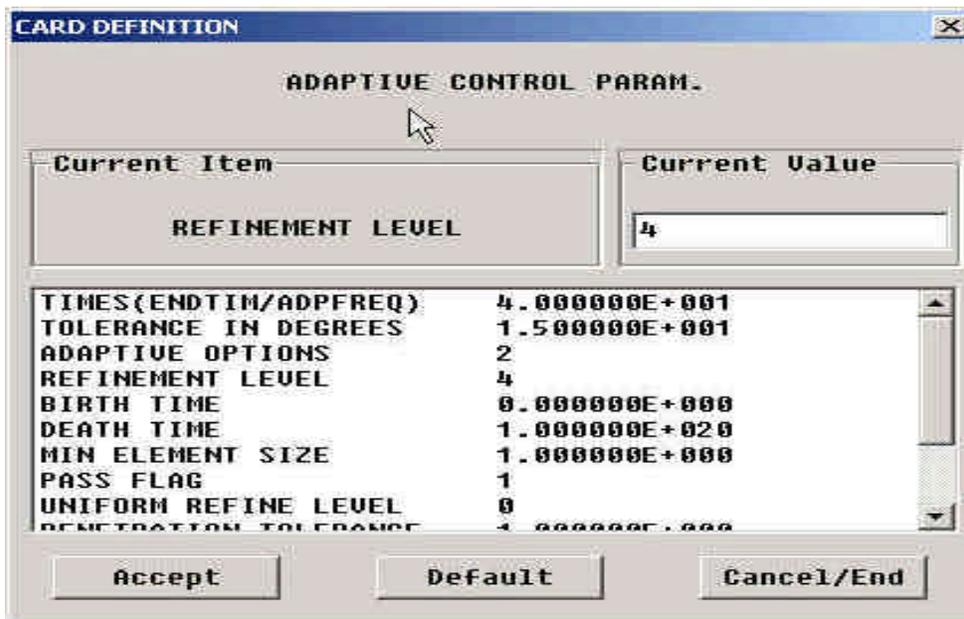
5. Now, click the **Control Parameters** button in the **Analysis Parameters** box.

6. As a new user it is recommended that you use the default control parameters. So, press **Default**, then **Accept**.
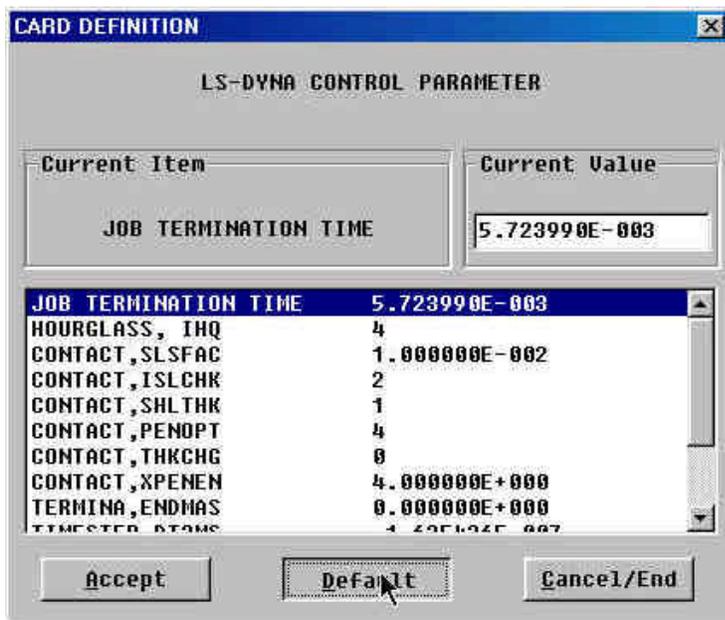
7. The job is ready to run. Be sure that **Full Run** is selected from the **Analysis Type** menu and press **Ok**. The Job will begin to run.
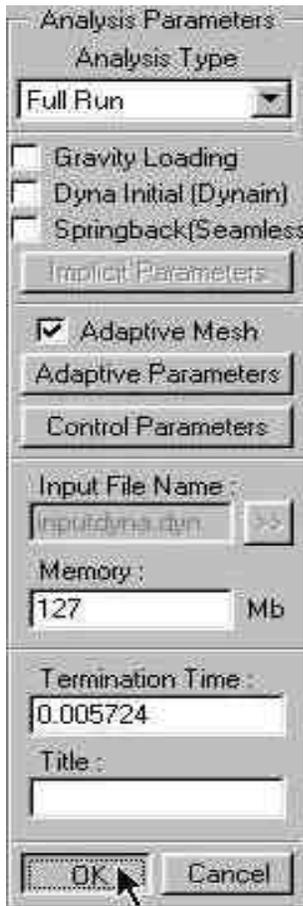
8. The solver is now running. You will notice that an estimated completion time is given. This time is not totally accurate since we are using adaptive mesh and the model will re-mesh several times. However, it does give you a ballpark idea.

9. Once the solver has given you the preliminary estimated time, you can refresh this value by first pressing **Ctrl-C**. This will momentarily pause the solver and prompt you to "**.enter sense switch:**" Then, enter the switch you would like to use and press enter:

> **sw1** – Terminates the Solver
>
> **sw2** – Refreshes the Estimated Solving Time
>
> **sw3** – Creates a d3dump Restart File
>
> **sw4** – Creates a d3plot File
>
> *Note:*   *These switches are case sensitive and must be all lower case when entered.*

Enter **sw2** and press **Enter**. Notice the estimated time has changed. You can use these switches at anytime while the solver is running.
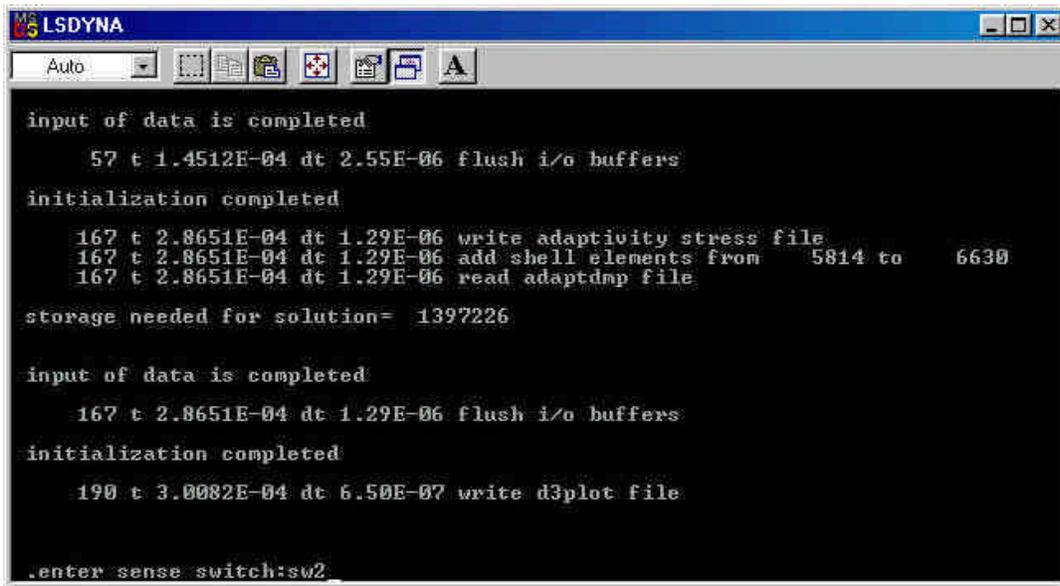
When you submit a job in Dynaform, an input deck is created which the solver, LS-DYNA, uses to process the results. The default input deck names are **lsdynainput.dyn** and **lsdynainput.mod.** The **.dyn** file contains all of the control cards, and the **.mod** file contains the geometry data. Advanced users are encouraged to study the **.dyn** input file. For more information, refer to the **LS-DYNA User's Manual** (LS-DYNA.pdf) located in C:\Program Files\eta\Dynaform\Manuals.

Note: *All files generated by either Dynaform or LS-DYNA will be placed in the directory in which the Dynaform database has been saved. This includes all input decks and post processing files.*

LS-DYNA Input File Generated using DYNAFORM-PC:

```
$ ETA/DYNAFORM : LS-DYNA(950) INPUT DECK
$ DATE  : Dec 18, 2000 at 16:36:16
$
$ VIEWING INFORMATION
$ -.171154E+02 .345210E+03-.119772E+03 .132504E+03
$  .707100E+00 .707100E+00 .000000E+00
$ -.353550E+00 .353550E+00 .866020E+00
$  .612370E+00-.612370E+00 .500000E+00
$
$ DRAW INFORMATION
$ DRAWTYPE DIRECTION  BEADTYPE  POSITION
$       2         3         1     1.000
$
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$
*KEYWORD
$
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$
$                   (1) TITLE CARD.
$
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
*TITLE
NONE
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$
$                   (2) CONTROL CARDS.
$
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$*CONTROL_STRUCTURED_TERM
*CONTROL_TERMINATION
$   ENDTIM    ENDCYC     DTMIN    ENDENG    ENDMAS
0.00572400        0                         .000
*CONTROL_TIMESTEP
$   DTINIT    TSSFAC      ISDO    TSLIMT     DT2MS      LCTM     ERODE
MS1ST
     .000      .900         0      .000-1.635E-07
*CONTROL_HOURGLASS
$      IHQ        QH
         4      .100
*CONTROL_BULK_VISCOSITY
$       Q1        Q2      TYPE
    1.500      .060         1
*CONTROL_SHELL
$   WRPANG    ITRIST     IRNXX    ISTUPD     THEORY       BWC     MITER
    20.000         2        -1         1         2         2         1
*CONTROL_CONTACT
$   SLSFAC    RWPNAL    ISLCHK    SHLTHK    PENOPT    THKCHG     ORIEN
```

```
       .010       .000          2          1          4          0          1
$  USRSTR    USRFAC    NSBCS    INTERM    XPENE    SSTHK    ECDT
TIEDPRJ
          0          0         10          0      4.000          0
*CONTROL_ENERGY
$   HGEN      RWEN    SLNTEN    RYLEN
          2          1          2          1
*CONTROL_DYNAMIC_RELAXATION
$  NRCYCK    DRTOL    DRFCTR    DRTERM    TSSFDR    IRELAL    EDTTL
IDRFLG
        250       .001       .995
*CONTROL_OUTPUT
$   NPOPT    NEECHO    NREFUP    IACCOP    OPIFS    IPNINT    IKEDIT
          0          0          0          0       .000          0        100
*CONTROL_PARALLEL
$    NCPU    NUMRHS     CONST
          1          0          2
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$
$                   (3) DATABASE CONTROL CARDS FOR ASCII FILE
$
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$*DATABASE_OPTION
$       DT
$
$OPTION : SECFORC RWFORC NODOUT ELOUT  GLSTAT
$         DEFORC  MATSUM NCFORC RCFORC DEFGEO
$         SPCFORC SWFORC ABSTAT NODFOR BNDOUT
$         RBDOUT  GCEOUT SLEOUT MPGS   SBTOUT
$         JNTFORC AVSFLT MOVIE
*DATABASE_MATSUM
5.7240E-05
*DATABASE_GLSTAT
5.7240E-05
*DATABASE_RCFORC
5.7240E-05
*DATABASE_SLEOUT
5.7240E-05
*DATABASE_RBDOUT
5.7240E-05
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$
$                   (4) DATABASE CONTROL CARDS FOR BINARY FILE
$
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
*DATABASE_BINARY_D3PLOT
$ DT/CYCL      LCDT    NOBEAM
3.0126E-04
*DATABASE_BINARY_RUNRSF
$ DT/CYCL      LCDT    NOBEAM
  .500E+04
*DATABASE_EXTENT_BINARY
```

```
$    NEIPH    NEIPS    MAXINT    STRFLG    SIGFLG    EPSFLG    RLTFLG
ENGFLG
                            5         1
$    CMPFLG   IEVERP    BEAMIP    DCOMP     SHGE      STSSZ
                  1                   2
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$
$                   (5) DEFINE TOOLS IN DYNAFORM
$
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$              BLANK          DESCRIPTION
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
*SET_PART_LIST
$SET_PART_NAME: BLANK
$       SID      DA1       DA2       DA3       DA4
          1
$     PID1      PID2      PID3      PID4      PID5      PID6      PID7
PID8
          2
*PART
$HEADING
 PART PID =        2 PART NAME :BLANK.LI
$       PID     SECID       MID     EOSID      HGID      GRAV     ADPOPT
          2         6         1                                       1
*MAT_3-PARAMETER_BARLAT
$MATERIAL NAME:BLANKMAT
$       MID        RO         E        PR        HR        P1        P2
          1 7.830E-09 2.070E+05 2.800E-01 2.000E+00 6.480E+02 2.300E-01
$         M       R00       R45       R90      LCID        E0       SPI
 8.000E+00 1.870E+00 1.270E+00 2.170E+00         0 0.000E+00 0.000E+00
$      AOPT
       2.0
$        XP        YP        ZP        A1        A2        A3
                                 1.000E+00 0.000E+00 0.000E+00
$        V1        V2        V3        D1        D2        D3
                                 0.000E+00 1.000E+00 0.000E+00
*SECTION_SHELL
$PROPERTY NAME:BLANKPRO
$     SECID    ELFORM      SHRF       NIP     PROPT   QR/IRID     ICOMP
          6         2   .830E+00       5.0       1.0        .0
$        T1        T2        T3        T4      NLOC
 1.000E+00 1.000E+00 1.000E+00 1.000E+00
*CONTROL_ADAPTIVE
$ ADPFREQ    ADPTOL    ADPOPT    MAXLVL    TBIRTH    TDEATH     LCADP
IOFLAG
1.4310E-04   .500E+01        2          30.0000e+001.0000E+20
1
$ ADPSIZE    ADPASS    IREFLG    ADPENE    ADPTH    MEMORY    ORIENT
MAXEL
     1.000         1         0     1.000      .500                   0
0
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$              PUNCH          DESCRIPTION
```

```
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
*SET_PART_LIST
$SET_PART_NAME: PUNCH
$        SID        DA1        DA2        DA3        DA4
           2
$       PID1       PID2       PID3       PID4       PID5       PID6       PID7
PID8
           5
*PART
$HEADING
 PART PID =         5 PART NAME :PUNCH
$        PID      SECID        MID      EOSID       HGID       GRAV     ADPOPT
           5          7          2
*MAT_RIGID
$MATERIAL NAME:PUNCHMAT
$       MID         RO          E         PR          N     COUPLE          M
ALIAS
           2 7.830E-09 2.070E+05 2.800E-01 0.000E+00 0.000E+00 0.000E+00
$       CMO       CON1       CON2
         1.0        4.0        7.0
$LCO or A1         A2         A3         V1         V2         V3

*SECTION_SHELL
$PROPERTY NAME:PUNCHPRO
$      SECID      ELFORM       SHRF        NIP      PROPT    QR/IRID      ICOMP
           7          2    .100E+01        3.0         .0         .0
$        T1         T2         T3         T4       NLOC
   .500E+00    .500E+00    .500E+00    .500E+00
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE
$        CID      CONTACT INTERFACE TITLE
$          2      BLANK/PUNCH
$       SSID       MSID      SSTYP      MSTYP     SBOXID     MBOXID        SPR
MPR
           1          2          2          2
$         FS         FD         DC         VC        VDC     PENCHK         BT
DT
   .125E+00    .000E+00    .000E+00    .000E+00    .200E+02
00.0000e+001.0000E+20
$        SFS        SFM        SST        MST       SFST       SFMT        FSF
VSF
   .000E+00    .000E+00    .000E+00    .000E+00
$       SOFT     SOFSCL     LCIDAB     MAXPAR     PENTOL      DEPTH      BSORT
FRCFRQ
           0
$     PENMAX     THKOPT     SHLTHK      SNLOG
                                           1
*BOUNDARY_PRESCRIBED_MOTION_RIGID
$        PID        DOF        VAD       LCID         SF        VID      DEATH
           5          3          0          1        1.0
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$              DIE            DESCRIPTION
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
*SET_PART_LIST
$SET_PART_NAME: DIE
```

```
$         SID          DA1          DA2          DA3          DA4
          3
$        PID1         PID2         PID3         PID4         PID5         PID6         PID7
PID8
          3
*PART
$HEADING
 PART PID =          3 PART NAME :DIE.S
$         PID        SECID          MID        EOSID         HGID         GRAV       ADPOPT
          3            8            3
*MAT_RIGID
$MATERIAL NAME:DIE   MAT
$         MID           RO            E           PR            N       COUPLE            M
ALIAS
          3 7.830E-09 2.070E+05 2.800E-01 0.000E+00 0.000E+00 0.000E+00
$         CMO         CON1         CON2
        1.0          7.0          7.0
$LCO or A1          A2           A3           V1           V2           V3

*SECTION_SHELL
$PROPERTY NAME:DIE   PRO
$       SECID       ELFORM         SHRF          NIP        PROPT      QR/IRID        ICOMP
          8            2    .100E+01          3.0           .0           .0
$          T1           T2           T3           T4         NLOC
   .500E+00     .500E+00     .500E+00     .500E+00
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE
$         CID        CONTACT INTERFACE TITLE
$          3        BLANK/DIE
$        SSID         MSID        SSTYP        MSTYP       SBOXID       MBOXID          SPR
MPR
          1            3            2            2
$          FS           FD           DC           VC          VDC       PENCHK           BT
DT
    .125E+00     .000E+00     .000E+00     .000E+00     .200E+02
00.0000e+001.0000E+20
$         SFS          SFM          SST          MST         SFST         SFMT          FSF
VSF
    .000E+00     .000E+00     .000E+00     .000E+00
$        SOFT       SOFSCL       LCIDAB       MAXPAR       PENTOL        DEPTH        BSORT
FRCFRQ
          0
$      PENMAX       THKOPT       SHLTHK        SNLOG
                                              1
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$               LOWER RING       DESCRIPTION
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
*SET_PART_LIST
$SET_PART_NAME: LOWER RING
$         SID          DA1          DA2          DA3          DA4
          5
$        PID1         PID2         PID3         PID4         PID5         PID6         PID7
PID8
          1
*PART
$HEADING
```

```
 PART PID =         1 PART NAME :BINDER.S
$       PID      SECID        MID      EOSID       HGID       GRAV     ADPOPT
         1          9          4
*MAT_RIGID
$MATERIAL NAME:LOWERMAT
$       MID         RO          E         PR          N     COUPLE          M
ALIAS
         4 7.830E-09 2.070E+05 2.800E-01 0.000E+00 0.000E+00 0.000E+00
$       CMO       CON1       CON2
       1.0        4.0        7.0
$LCO or A1         A2         A3         V1         V2         V3

*SECTION_SHELL
$PROPERTY NAME:LOWERPRO
$     SECID      ELFORM       SHRF        NIP      PROPT    QR/IRID      ICOMP
         9          2    .100E+01        3.0         .0         .0
$        T1         T2         T3         T4       NLOC
   .500E+00    .500E+00    .500E+00    .500E+00
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE
$       CID    CONTACT INTERFACE TITLE
$         5    BLANK/LOWER RING
$      SSID       MSID      SSTYP      MSTYP     SBOXID     MBOXID        SPR
MPR
         1          5          2          2
$        FS         FD         DC         VC        VDC     PENCHK         BT
DT
   .125E+00    .000E+00    .000E+00    .000E+00    .200E+02
00.0000e+001.0000E+20
$       SFS        SFM        SST        MST       SFST       SFMT        FSF
VSF
   .000E+00    .000E+00    .000E+00    .000E+00
$      SOFT     SOFSCL     LCIDAB     MAXPAR     PENTOL      DEPTH      BSORT
FRCFRQ
         0
$    PENMAX     THKOPT     SHLTHK      SNLOG
                                          1
*LOAD_RIGID_BODY
$       PID        DOF       LCID         SF        CID         M1         M2
M3
         1          3          2        1.0
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$
$                  (7) DEFINE CURVE
$
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$---------TRAVELING LOAD CURVE DATA FOR PUNCH
*DEFINE_CURVE
$CURVENAME  PUNTRV1         10
$      LCID       SIDR       SCLA       SCLO       OFFA       OFFO
         1          0
$                   A1                 O1
     .000000000E+00      .000000000E+00
     .806996600E-04      .125147600E+04
     .161399300E-03      .247213600E+04
     .242099000E-03      .363192400E+04
```

```
         .322798600E-03          .470228200E+04
         .403498300E-03          .565685400E+04
         .484198000E-03          .647213600E+04
         .564897600E-03          .712805200E+04
         .645597300E-03          .760845200E+04
         .726297000E-03          .790150700E+04
         .806996700E-03          .800000000E+04
         .491699600E-02          .800000000E+04
         .499769600E-02          .790150700E+04
         .507839500E-02          .760845200E+04
         .515909500E-02          .712805200E+04
         .523979400E-02          .647213600E+04
         .532049400E-02          .565685400E+04
         .540119400E-02          .470228200E+04
         .548189300E-02          .363192400E+04
         .556259300E-02          .247213600E+04
         .564329200E-02          .125147600E+04
         .572399200E-02          .000000000E+00
$---------FORCE LOAD CURVE DATA FOR LOWER RING
*DEFINE_CURVE
$CURVENAME  LOWFOR2          11
$      LCID       SIDR       SCLA       SCLO       OFFA       OFFO
          2          0
$                    A1                    O1
       .000000000E+00       .000000000E+00
       .286200000E-03       .200000000E+06
       .572400000E-02       .200000000E+06
$---------OTHER CURVES
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$
$       MODEL DATA IN FILE  :inputdyna.mod
$       NUMBER OF NODES     :     5475
$       NUMBER OF SHELLS    :     5511
$
*INCLUDE
inputdyna.mod
$
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+--
--8
$
*END
```

IX. Post Processing

DYNAFORM-PC can read and process all the available data in the **d3plot** file. In addition to the undeformed model data, the **d3plot** file also contains all result data generated by LS-DYNA (stress, strain, time history data, deformation, etc.).

Reading the d3plot File into the Post Processor

Start the Dynaform-PC Post Processor, PostGL. The default path for PostGL is C:\Program Files\eta\Dynaform. In this directory, double click the executable file, **PostGL.exe**. The post processor will be started.

1. From the **File Menu**, select **Open**.
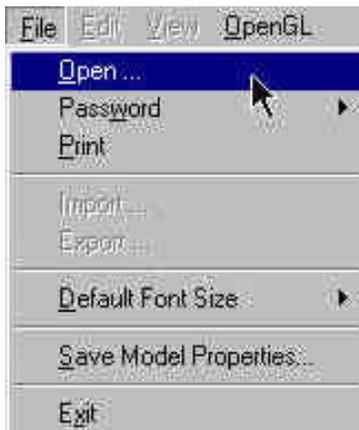
The Open File box will be displayed.
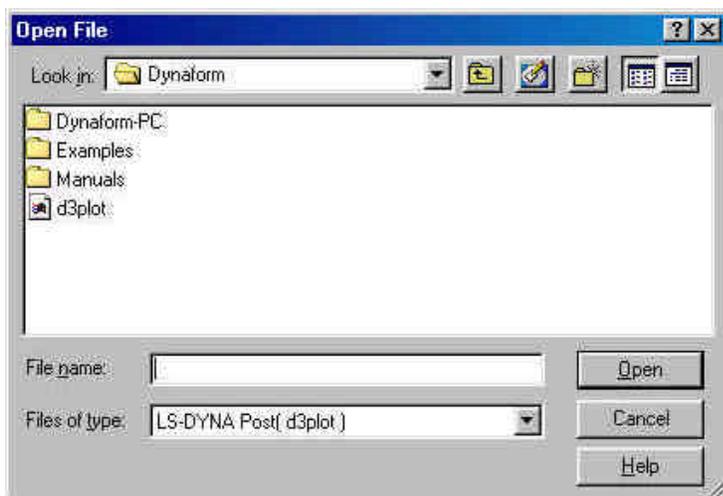
2. Select **LS-DYNA Post (d3plot) to PP file** from the **Files of Type** list. This option will allow you to read in the d3plot file and then automatically create a file called **PostGL.pp**. The **.pp** file can be read in much faster then the d3plot file, and allows you to save space, since the **.pp** file is much smaller then the d3plot files. You will only need to read in the d3plot file once. From then on, you should use the **PostGL.pp** file.

   After moving to the directory where you saved the Dynaform database, be sure you have the correct file of type selected, pick the **d3plot** file, and press **Open**.
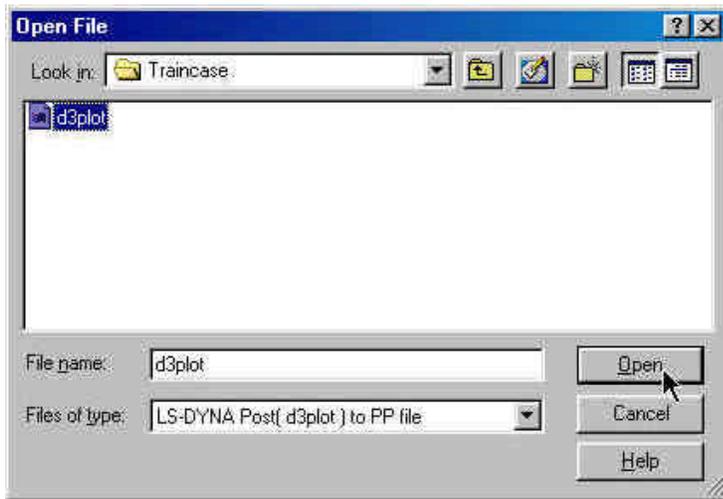
3. When prompted to "**SELECT LS-DYNA VERSION**", select **LS940/LS950** and press **Ok**.

4. When prompted to "**SELECT TIME STEPS**", select **ALL AVAILABLE STEPS** and press **Ok**.


5. When prompted to "**SELECT RESULT COMPONENTS**", select **ALL ITEMS** then and press **Ok**.
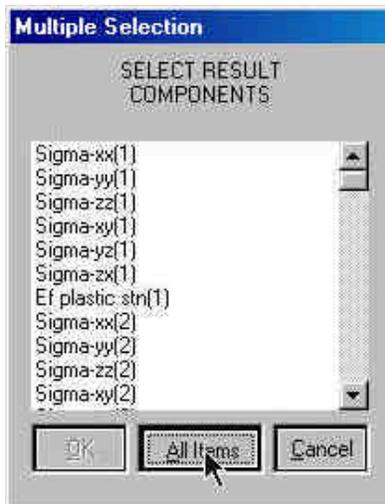
6. The d3plot file is now completely read in. You are ready to animate the results.

Animating Deformation

1. In the **Plot State** dialogue box, select **Deformation**. In the **Frame Range** box select **All Frames** and then press **Play**. The results will be animated.
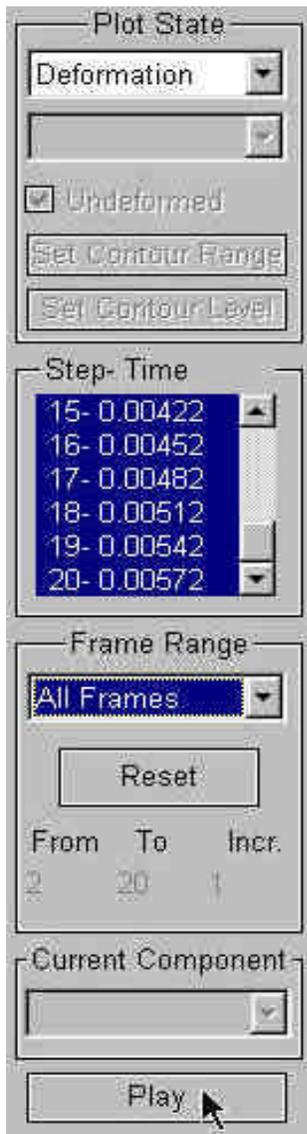

2. Shade the results by pressing the **Shade Model** button near the bottom of the screen.

3. Since it is difficult to see the Blank with all of the other tools displayed, you can turn them all off, leaving only the Blank showing. In the **Control Options** box, select **OFF/ON by Name**.
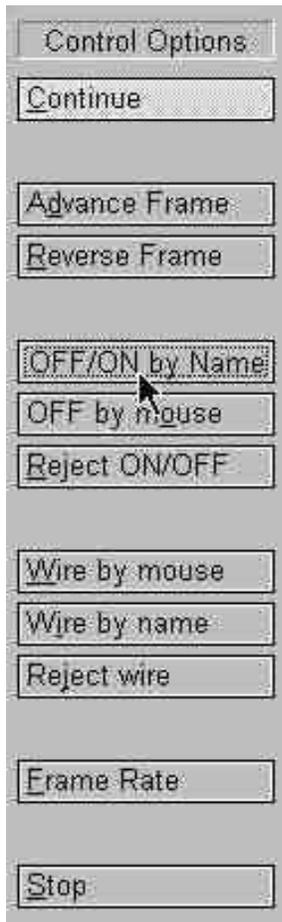
4. Turn off all of the parts except for the Blank and press **Ok**.
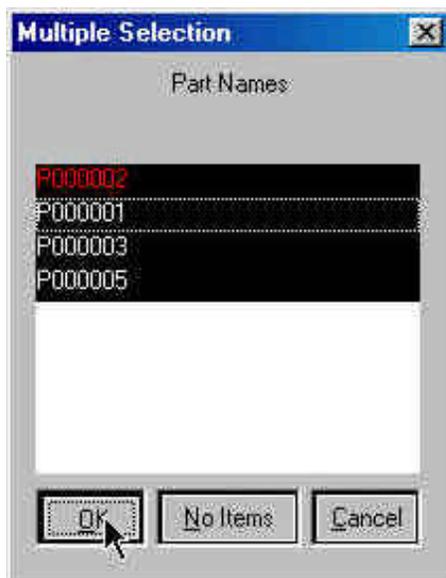
5. You can also change the view with the view manipulation icons on the tool bar, just as we did in the pre processor.

View Manipulation

6. Try adjusting the frame rate by pressing the **Frame Rate** button in the **Control Options** window. Sometimes this is necessary is the animation moves too quickly.
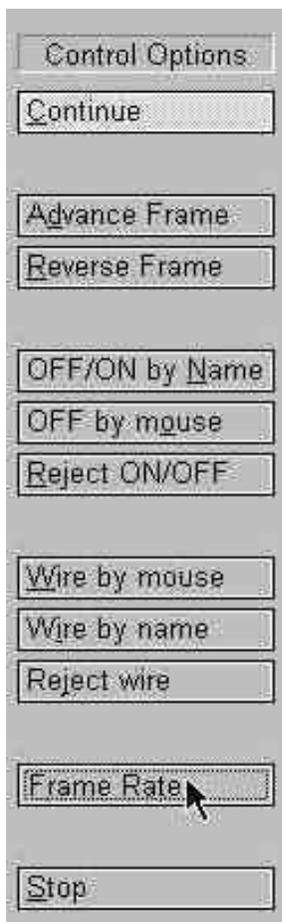
7. You will be prompted to enter a new frame rate. Enter **5** and press **Accept**.

| Frame/Second	Cancel/End
5	Accept

Notice the change in the speed of the animation. Adjust the frame rate until you find it acceptable.

8. Another way to control the speed of the animation is to advance or reverse frames manually. To do this, press either the Advance Frame or Reverse Frame button in the Control Options Box. Then move the mouse pointer into the display area and left-click. Notice the frames will advance, or reverse, one at a time.

Control Options
Continue
Advance Frame
Reverse Frame
OFF/ON by Name
OFF by mouse
Reject ON/OFF
Wire by mouse
Wire by name
Reject wire
Frame Rate
Stop
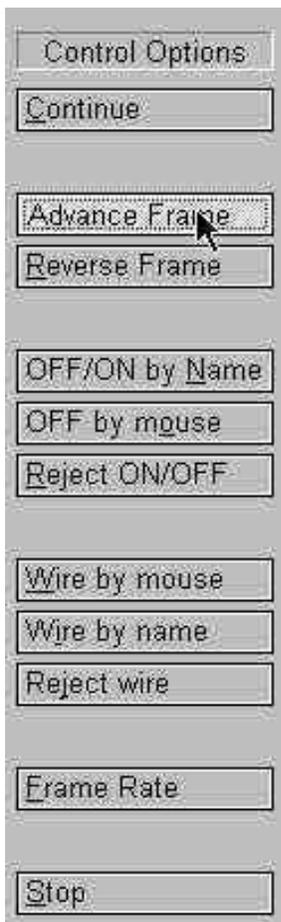
9. When you are done viewing the animation, press **Stop** in the **Control Options** box.

Control Options

Continue

Advance Frame
Reverse Frame

OFF/ON by Name
OFF by mouse
Reject ON/OFF

Wire by mouse
Wire by name
Reject wire

Frame Rate

Stop
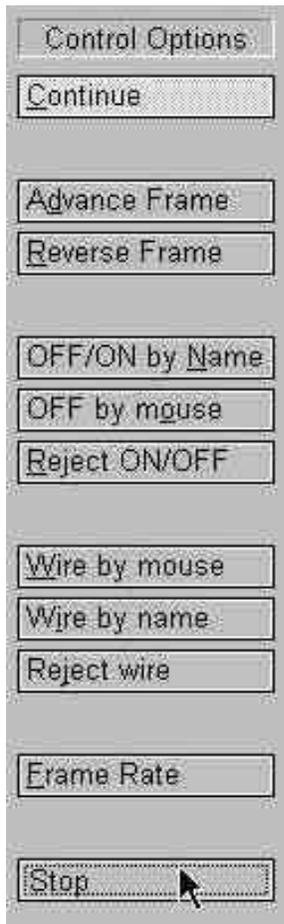
Animating Stresses/Strains and FLD

PostGL can also animate various stresses and strains as well as FLD. To do this, refer to the following examples.

Stress/Strain

1. Pick **Contour** and **Stress/Strain** from the **Plot State** box.
2. Select **All Frames** from the **Frame Range** menu.
3. Select the type of **Stress/Strain** you would like to view from the **Current Component** list. (The picture below has Thickness selected)
4. Press **Play** and the animation will begin.
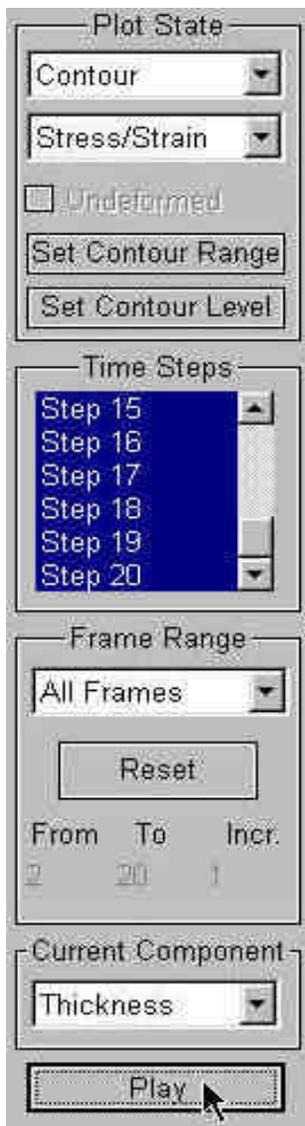
FLD

1. Pick **FLD** and **On Element** from the **Plot State** box.
2. Select **All Frames** from the **Frame Range** menu.
3. Select **Middle** from the **Current Component** list.
4. Press **Play** and the animation will begin.
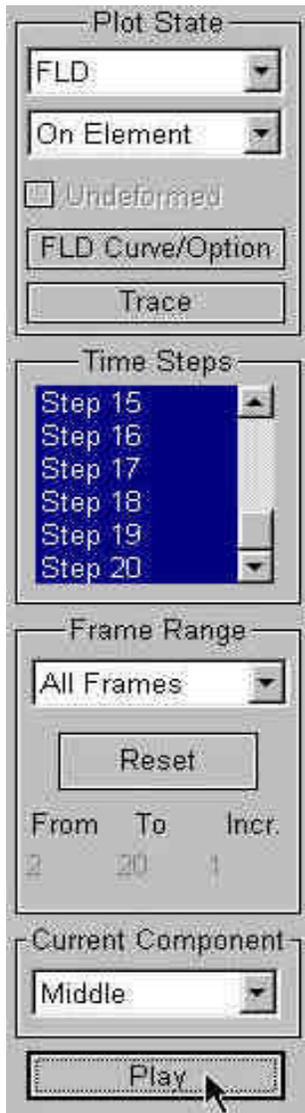
Plotting Single Frames

It is sometimes more convenient to view single frames rather than the entire animation To do this, select **Single Frame** from the **Frame Range** box and then select, with your mouse, the frame you would like to view. Refer to the example below in which the FLD plot of the 17th frame would be displayed upon pressing **Plot**.
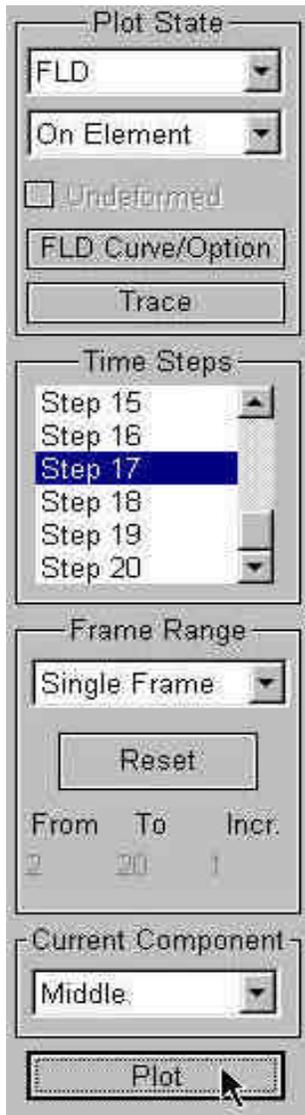
Writing an AVI File

PostGL has a very useful tool that allows you to automatically create an avi movie via an animation screen capture. This will be the last function covered in this training case.

1. Start a new animation using all available steps.
2. Once the animation is running, click on the **Write AVI File** icon, located on the **Toolbar**.

3. The **Write File** box will be displayed. Enter a name to save the avi file under (e.g. traincase.avi), and press **Save**.
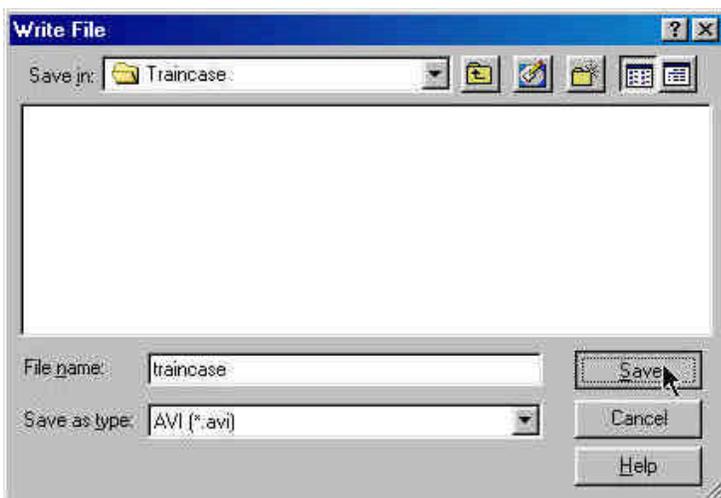
4. Enter a frame rate in the prompt box. It is recommended to use the default. Press **Accept**.


5. Enter a window size in the prompt box. It is recommended to use the default. Press **Accept**.

6. Select **Build** from the menu bar.

7. Select **Microsoft Video 1** from the **Compressor** list and press **Ok**.
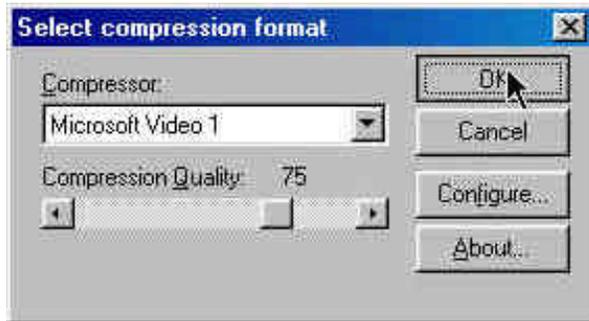
8. PostGL will now take a screen capture of the animation and write the output.

X. Conclusion

This concludes the training guide's basic overview of Dynaform-PC. You should now have the basic skills necessary to create models, prepare models for analysis, and display results. Please reference the User's Manual for a more detailed explanation of each individual function.