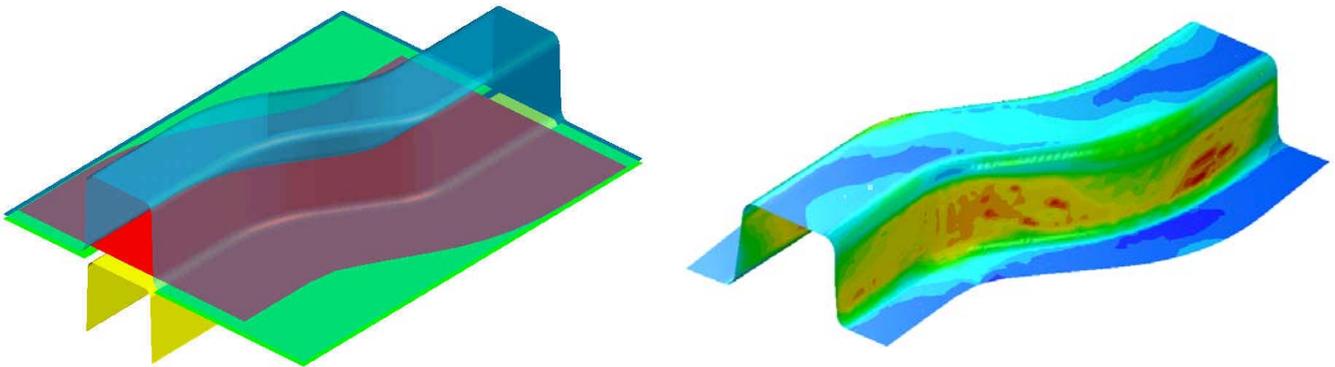


# eta/DYNAFORM

## *Training Manual*



### Version 5.5

Engineering Technology Associates, Inc.  
1133 E. Maple Road, Suite 200  
Troy, MI 48083

Tel: (248) 729-3010  
Fax: (248) 729-3020  
Email: [support@eta.com](mailto:support@eta.com)

eta/DYNAFORM team  
August 2006

Engineering Technology Associates, Inc. All rights reserved ©1998-2006.

# FORWARD

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design.

This material is a compilation of data and figures from many sources.

Engineering Technology Associates, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

# INTRODUCTION

Welcome to the **eta/DYNAFORM version 5.5** Training Manual. The **eta/DYNAFORM version 5.5** is the unified version of the DYNAFORM-PC and UNIX platforms. This manual is meant to give the user a basic understanding of finite element modeling for forming analysis, as well as displaying the forming results. It is by no means an exhaustive study of the simulation techniques and capabilities of **eta/DYNAFORM**. For more detailed study of **eta/DYNAFORM**, the user is urged to attend an **eta/DYNAFORM** training seminar.

This manual details a step-by-step sheet metal forming simulation process through traditional finite element (FE) modeling and the newer QuickSetup interface process. Users should take the time to learn both setup processes as each has inherent benefits and limitations.

The traditional setup procedure is extremely flexible and can be used to setup any forming simulation. Because the QuickSetup makes certain assumptions as to the type of tooling configuration selected it is not as flexible, however, it automates many of the procedures required for traditional model setup such as travel curves.

The following table outlines the major differences between the QuickSetup and the traditional setup procedure.

TRADITIONAL SETUP	QUICK SETUP	AUTO SETUP
Manual interface can duplicate any tooling configuration: pads, multiple tools, etc.	Automated interface limits flexibility	There are some inner templates that enables user to setup all kinds of operation.
Requires more setup time	Reduces modeling setup time	Reduces setup time and reduces the possibility of make mistake
Manual definition of travel curves	Automated travel curves	Automated travel curves and manual definition curve
Geometrical Offset and Contact Offset	Contact Offset	Both Contact Offset and Geometrical Offset

*Note: This manual is intended for the application of all **eta/DYNAFORM** platforms. Platform interfaces may vary slightly due to different operating system requirements. This may cause some minor visual discrepancies in the interface screen shots and your version of **eta/DYNAFORM** that should be ignored.*

# TABLE OF CONTENTS

<b>DATABASE MANIPULATION .....</b>	<b>1</b>
I. CREATING AN ETA/DYNAFORM DATABASE AND ANALYSIS SETUP .....	1
II. PRACTICE SOME AUXILIARY MENU OPERATIONS .....	4
III. TURNING PARTS ON/OFF .....	5
IV. EDITING PARTS IN THE DATABASE .....	7
V. CURRENT PART .....	9
<b>MESHING .....</b>	<b>11</b>
I. Blank Meshing .....	11
II. Meshing Surface Data .....	14
III. Mesh Checking .....	17
IV. QuickSetup vs. Traditional Setup .....	22
<b>QUICKSETUP .....</b>	<b>24</b>
I. Define the Lower Ring from the Lower Tool .....	24
II. QuickSetup interface: .....	29
III. Define Tools .....	30
IV. Defining the Blank Material .....	33
V. Running the Analysis .....	37
<b>AUTO SETUP .....</b>	<b>43</b>
I. New Simulation .....	43
II. General .....	45
III. Blank Definition .....	46
IV. Tools Definition .....	49
V. Tool Positioning .....	58
VI. Process Definition .....	61
VII. Animation .....	63
VIII. Submit Job .....	65
<b>TRADITIONAL SETUP .....</b>	<b>68</b>

I. Offsetting the Lower Tool Mesh to Create the Upper Tool .....	68
II. Create the Lower Ring .....	71
III. Separate LOWRING and LOWTOOL .....	75
IV. Draw Type Setup.....	76
V. Tool Definition.....	77
VI. Defining the Blank and Setting up Processing Parameters.....	80
VII. Tools Summary .....	84
VIII. Auto Positioning the Tools.....	85
IX. Measuring the DIE Travel Distance.....	87
X. Define DIE Velocity Curve.....	89
XI. Defining the Binder (LOWRING) Force Curve .....	92
XII. Preview Tool Animation .....	94
XIII. Running the Analysis .....	95
<b>POST PROCESSING (with eta/POST) .....</b>	<b>98</b>
I. Reading the Results File into the Post Processor.....	98
II. Animating Deformation .....	100
III. Animating Deformation, Thickness and FLD .....	101
IV. Plotting Single Frames.....	104
V. Writing an AVI and E3D File.....	105
<b>MORE ABOUT DYNAFORM 5.5 .....</b>	<b>108</b>
<b>CONCLUSION.....</b>	<b>109</b>

# DATABASE MANIPULATION

## I. CREATING AN ETA/DYNAFORM DATABASE AND ANALYSIS SETUP

### Start eta/DYNAFORM 5.5.

For workstation/Linux users, enter the command “**df55**” (default) from a UNIX shell.  
 For PC users, double click the **eta/DYNAFORM 5.5 (DF55)** icon from the desktop.

After starting **eta/DYNAFORM**, a default database **Untitled.df** is created. You will begin by importing CAD files to the current database.

### Import files

1. From the menu bar, select **File**→**Import**. to display the Import file dialog window illustrated in Figure 1.

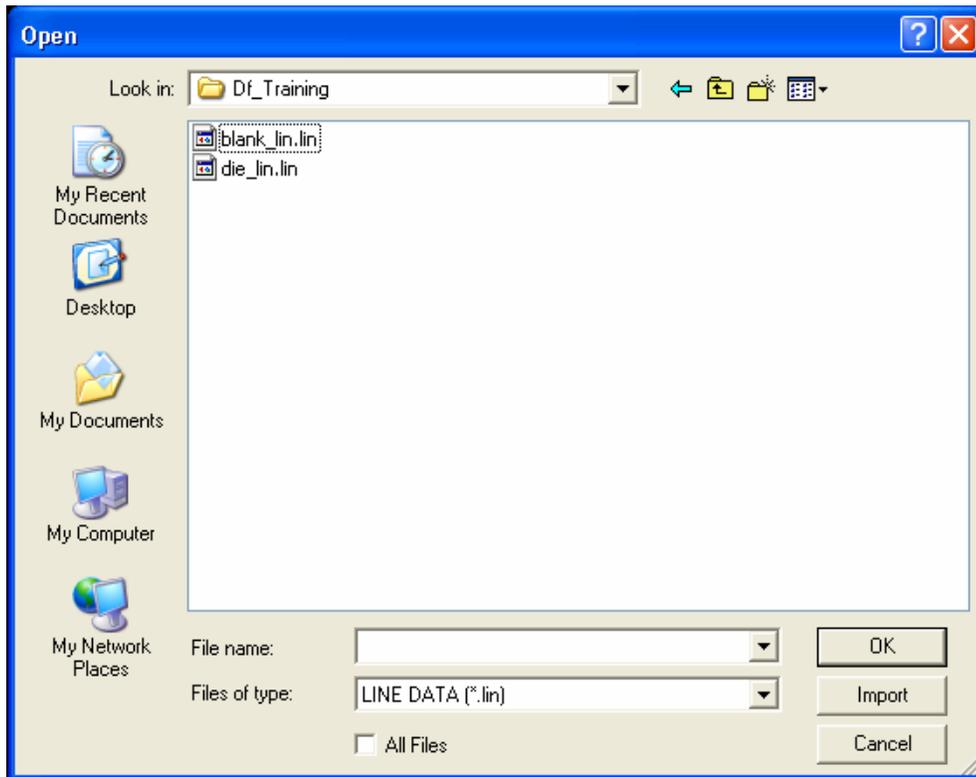


Figure 1: Import file dialog window

Click the drop-down button of **Files of type** to change the file format to “**LINE DATA (\*.lin)**”.  
 Go to the training input files located in the CD provided along with the eta/DYNAFORM

installation. Locate two data files: **die.lin** and **blank.lin**. Then, import both files and select **OK** to dismiss the **Import File** dialog window.

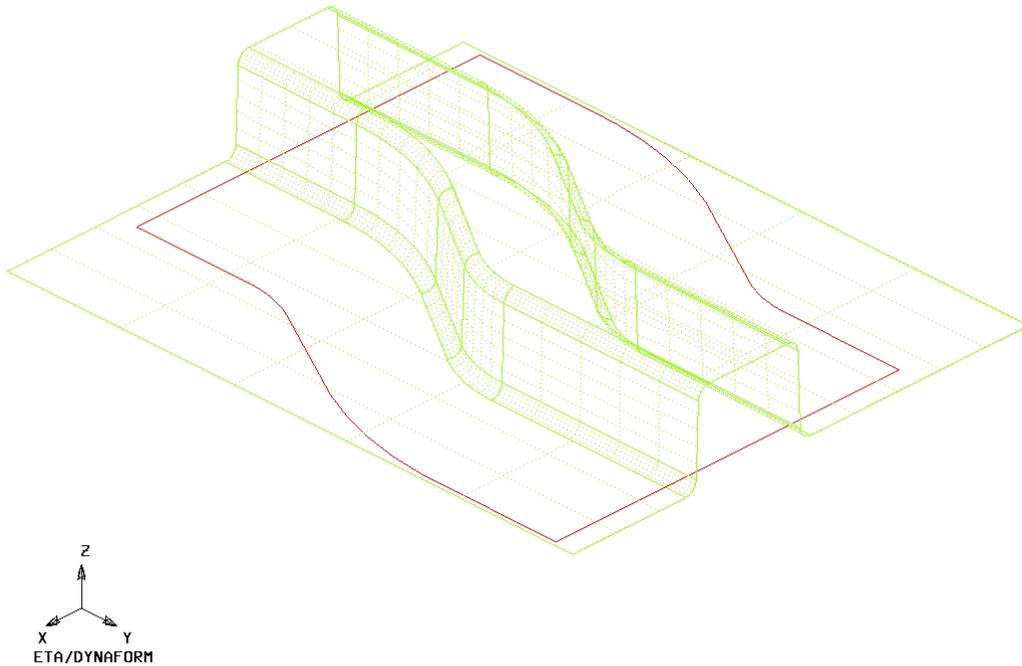


Figure 2: Illustration of S-Rail model

After reading in all of the data files, verify the display looks the same as the illustration shown Figure 2. The parts are displayed in the isometric view which is the default view setting of eta/DYNAFORM.

**Note:** *Icons may appear different depending on platform. Other functions on the **Icon bar** will be discussed further in the next section. You can also refer to the **eta/DYNAFORM User's Manual** for information on all of the **Icon bar** functions.*

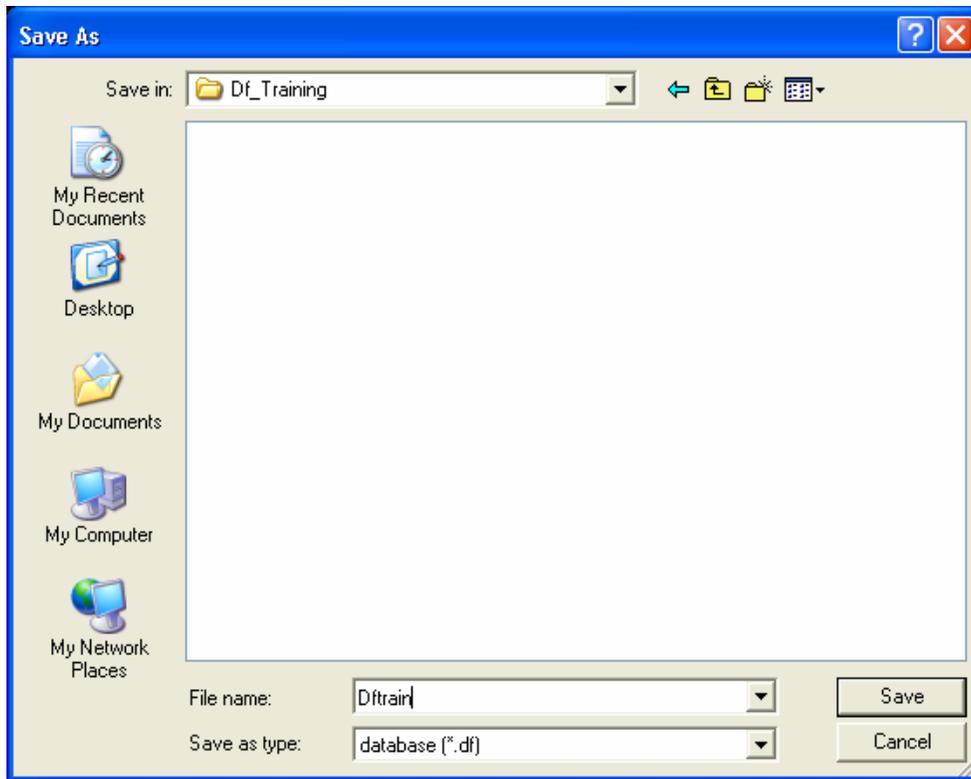


Figure 3: Save as dialog window

2. Save the database to the designated working directory. Go to **File**⇒**Save as** to open the dialog window illustrated in Figure 3. Then, type “**dftrain.df**” in the File name input field, and select **Save** to dismiss the dialog window.

### Analysis Setup

The default unit system of a new eta/DYNAFORM database is **mm, Newton, second, and Ton**. The default setting for draw type is **double action (toggle draw)**. You may change these default settings from the **Tools**⇒**Analysis Setup** menu.

***Notice: Draw Type** corresponds with **press type** in practice. The parameters define the working direction of default punch and die. If you are not sure of operating process, you should select **User Defined**. You can also refer to the **eta/DYNAFORM User’s Manual** for information on all of the **Draw Type** functions.*

### File Types

eta/DYNAFORM has the ability to read the following types of input files:

- |                                |                                      |
|--------------------------------|--------------------------------------|
| 1. Abaqus (*.inp)              | 8. VDA (*.vda,vdas)                  |
| 2. LS-DYNA (*.dyn, *.mod, *.k) | 9. DYNAIN file (dynain*, *.din)      |
| 3. NASTRAN (*.nas, *.dat)      | 10. CATIA v4/v5 (*.model, *.CADPart) |
| 4. Stereo lithograph (*.stl)   | 11. ProE (*.prt, *.asm)              |

- |                         |                         |
|-------------------------|-------------------------|
| 5. Autocad (*.dxf)      | 12. STEP (*.stp)        |
| 6. Line Data (*.lin)    | 13. Unigraphics (*.prt) |
| 7. Iges (*.igs, *.iges) | 14. dynain file         |

## II. PRACTICE SOME AUXILIARY MENU OPERATIONS

After successfully reading in the needed files to begin the finite element modeling, practice to get familiar with some of the basic functions and menus.

### View Manipulation



#### View Manipulation

Figure 4: View manipulation

The view manipulation area of the **Icon bar** (shown in Figure 4) is one of the most visited spots in eta/DYNAFORM. These functions enable you to change the orientation of the display area. Place the mouse pointer over each icon to display the name and function of each icon. Also, take notice of the **Display Options** area (shown in Figure 5) at the bottom right corner of the screen. This is another area enabling you to manipulate the display area.



Figure 5: Display options dialog

The following steps will help you to become more familiar with the functions found in the **Icon bar**, and in the **Display Options** dialog.

1. Select **Isometric** from the **Icon bar**. This places the displayed geometry in an isometric view, as shown earlier. The function is shown in Figure 6:



Figure 6: Isometric icon

2. Rotate the geometry dynamically about the z-axis approximately 90° by using the **Rotate about Z-Axis** function. The function is shown in Figure 7:



Figure 7: Rotate about Z-axis icon

3. Select **Right View (YZ Plane)**. The result is shown in Figure 8:



Figure 8: Illustration of model

4. Select **Free Rotation** to rotate the model.



Figure 9: Free rotate icon

5. Select **Fill Screen** from the **Icon bar**, this makes the displayed geometry fill the screen.



Figure 10: Fill screen icon

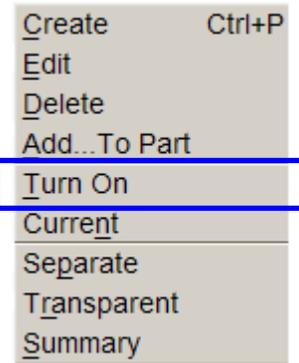
6. Practice using the other viewing options before moving on.

### III. TURNING PARTS ON/OFF

All geometry in eta/DYNAFORM is arranged based on parts. Every entity, by default, will be created or read into a part. You should practice using the **On/Off** function, located in the **Icon bar**, or in the Parts menu: **Parts**⇒**Turn On**. See Figure 11.



(a)



(b)

Figure 11: Part on/off function in (a) Icon bar (b) Menu

1. Notice the **Turn On/Off Part** dialog box that appears after selecting the **Part On/Off** button.

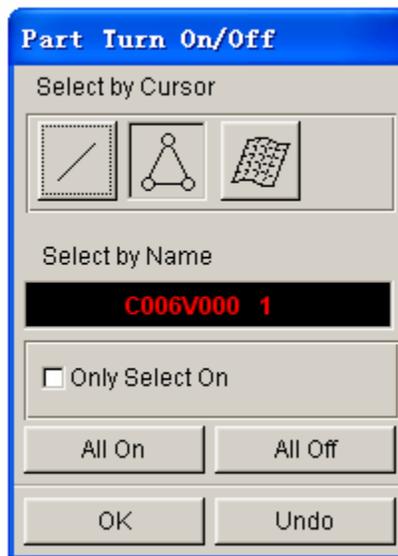


Figure 12: Part turn on/off dialog box

Place the cursor over different icons and buttons to learn the name of each function. This type of selection menu is common in the eta/DYNAFORM environment. It provides several different functions for selecting which parts will be turned off or on.

2. Since the part **BLANK.LIN** contains only line data, you will have to use either the **Select by Line** or the **Select by Name** function.
3. First, use the **Select by Line** icon to turn off the part, **BLANK.LIN**. Click the **Select by Line** icon, and select a line in the part, **BLANK.LIN**. The part will be turned off.

4. Next, use the **Select by Name** function, and select **BLANK.LIN** from the list in the dialog box. In the **Select by Name** field, the parts that are turned on are displayed in their color and the parts that are turned off are displayed in white.
5. Before we continue, verify that all available parts are turned on. Select the **All On** button from the **Turn On/Off Part** dialogue.
6. After you have turned all the parts on, click the **OK** button on the **Turn On/Off Part** dialog. This will end the current operation.

#### IV. EDITING PARTS IN THE DATABASE

The **Edit Part** command, illustrated in Figure 13, is used to edit part properties and delete parts.

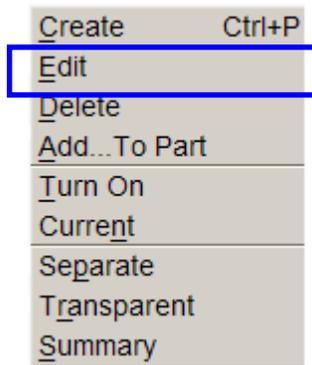


Figure 13: Edit part command

1. As illustrated in Figure 13, select the **Edit** function.

The **Edit Part** dialog box shown in Figure 14 will be displayed, with a list of all the parts that are defined in the database. The parts are listed with the part name and identification number. From here, you can modify the part name, ID number and part color. You can also delete parts from the database.

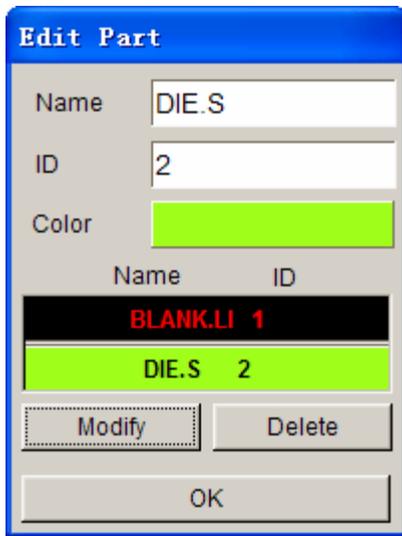


Figure 14: Edit part dialog box

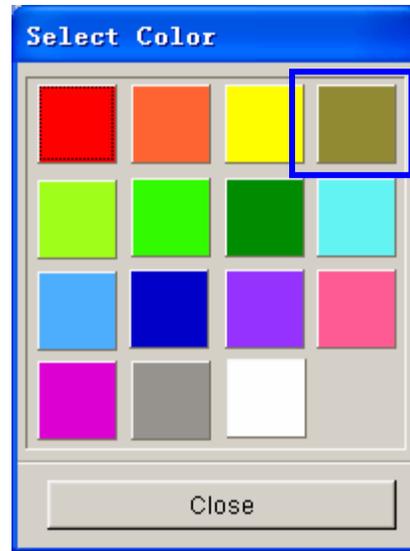


Figure 15: Select color dialog box

2. Select the part **DIE.S** from the part list. Change its color by clicking on the Color button and selecting a new color from the palette illustrated in Figure 15.
3. Select the part **DIE.S** from the part list. Change the **Name** by inputting **LOWTOOL** in the field as shown below.

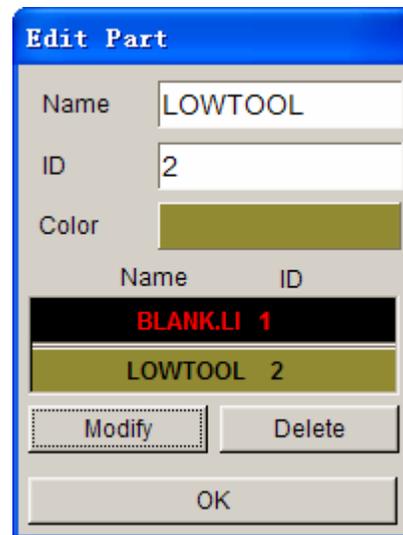
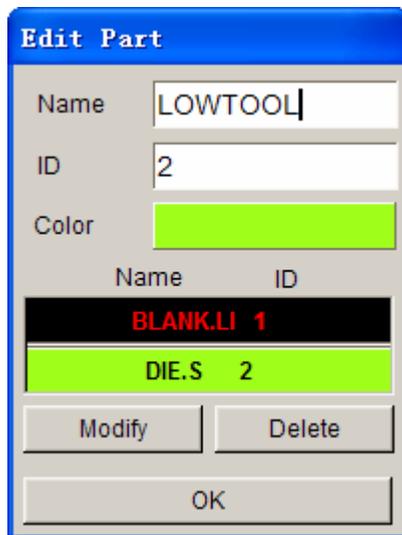


Figure 16: Editing part name and color

4. Once you have entered the new name (**LOWTOOL**) and selected the desired part color, click on the **Modify** button located at the bottom left of the dialog to complete the changes.
5. Click **Close** to end this operation.

6. Save your database.

*Note:* Designers often model only one surface of upper or lower die. The other die face will offset from the mating surface. In this case, we suppose the surface is the lower tool. Hence, We choose the part name as **LOWTOOL**. The upper tool will offset from the surface.

## V. CURRENT PART

All lines, surfaces, and elements which were created 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 on.

1. To change the current part, click on the **Current Part** icon in the **Display Options** dialog illustrated in Figure 17.



Figure 17: Current part icon

Or select **Parts**→**Current** on **Menu bar**

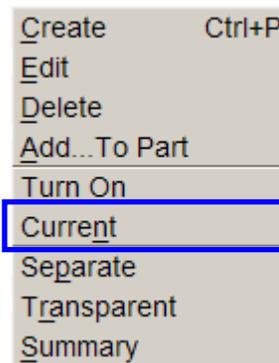


Figure 18: Current part function

2. The **Current Part** dialog box will be displayed.

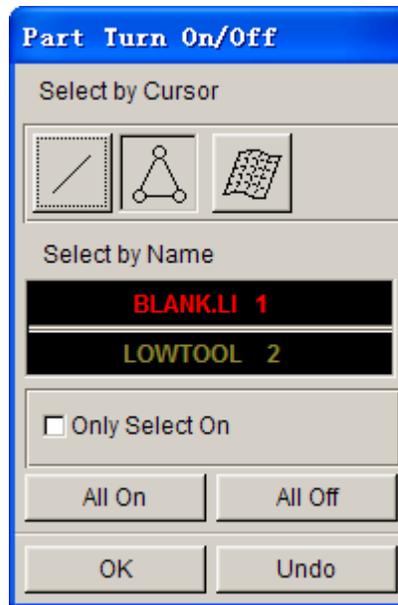


Figure 19: Part turn on/off dialog box

3. Similar to the **Part Turn On/Off** dialog box, you may select the current part in different ways. Place the cursor over each icon to identify its function.
4. Set the part **BLANK.LI** as current by selecting the part name from the **Select by Name** list that is displayed.
5. Practice setting the current part.
6. Turn off all of the parts except **BLANK.LI**, and set it as current.

# MESHING

Meshing from surface or line data is a very important step contributing to a successful simulation. There are many methods of creating mesh, however in this exercise, the Blank Generator and Surface Mesh will be used to generate meshes.

## I. BLANK MESHING

Blank meshing is the most important mesh function since the accuracy of the forming results depends heavily upon the quality of blank mesh. There is a special function for blank meshing.

1. Select **Tools**→**Blank Generator** on **Menu bar**, as illustrated in Figure 20.

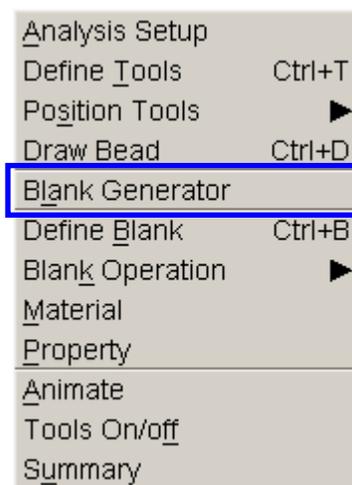


Figure 20: Tools menu

2. Select the **BOUNDARY LINE** option from the **Select Option** dialog box illustrated in Figure 21.

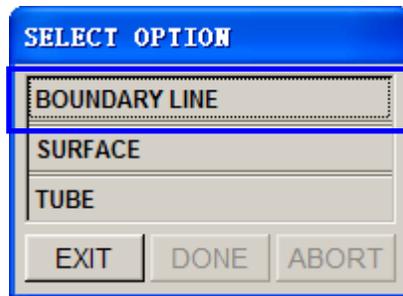


Figure 21: Select option dialog box

3. The **Select Line** dialog box shown in Figure 22 is displayed.

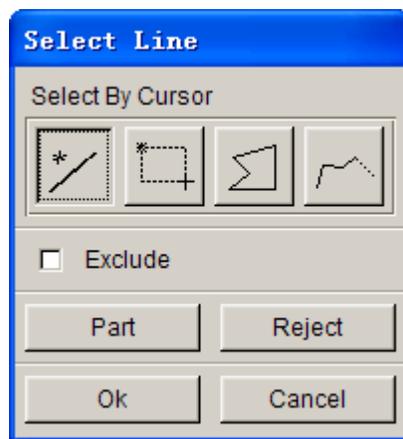


Figure 22: Select line dialog box

The **BLANK.LI** part consists of 4 line segments. Select the lines one-by-one by first placing your mouse cursor on each one of them, following by clicking on left mouse button. The **Select Line** dialog box allows you to select the line(s) in different ways. Place the cursor over each icon and button to identify its function.

4. Select **OK** after selecting all four lines.
5. Use the default variable (6.0) for the concerned tool radii. This number reflects the tightest concerned die radii in the model. The smaller the radii, the finer the blank mesh; a larger value will result in coarser mesh.

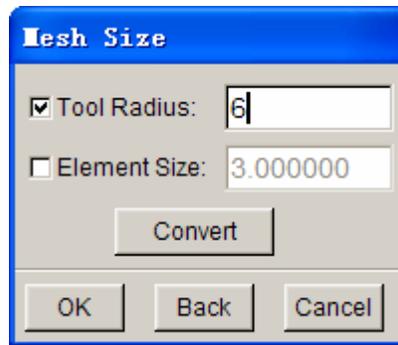


Figure 23: Concerned tool radius

6. After you have entered the variable, press **OK**. A **Dynaform Question** dialog box pops up asking you to accept, deny or remesh the mesh. Press **Yes** to accept the mesh.

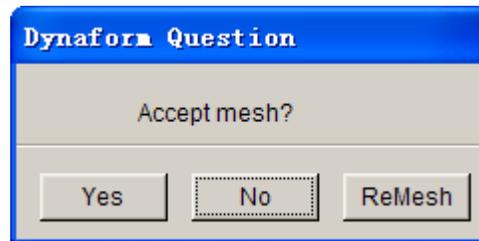


Figure 24: Dynaform Question dialog box

If you have entered an incorrect value, click on the **ReMesh** button to correct the concerned tool radii value and accept the mesh, or **No** to cancel it and repeat the above steps to re-mesh.

7. Compare your mesh with the picture shown in Figure 25.
8. Save the database.

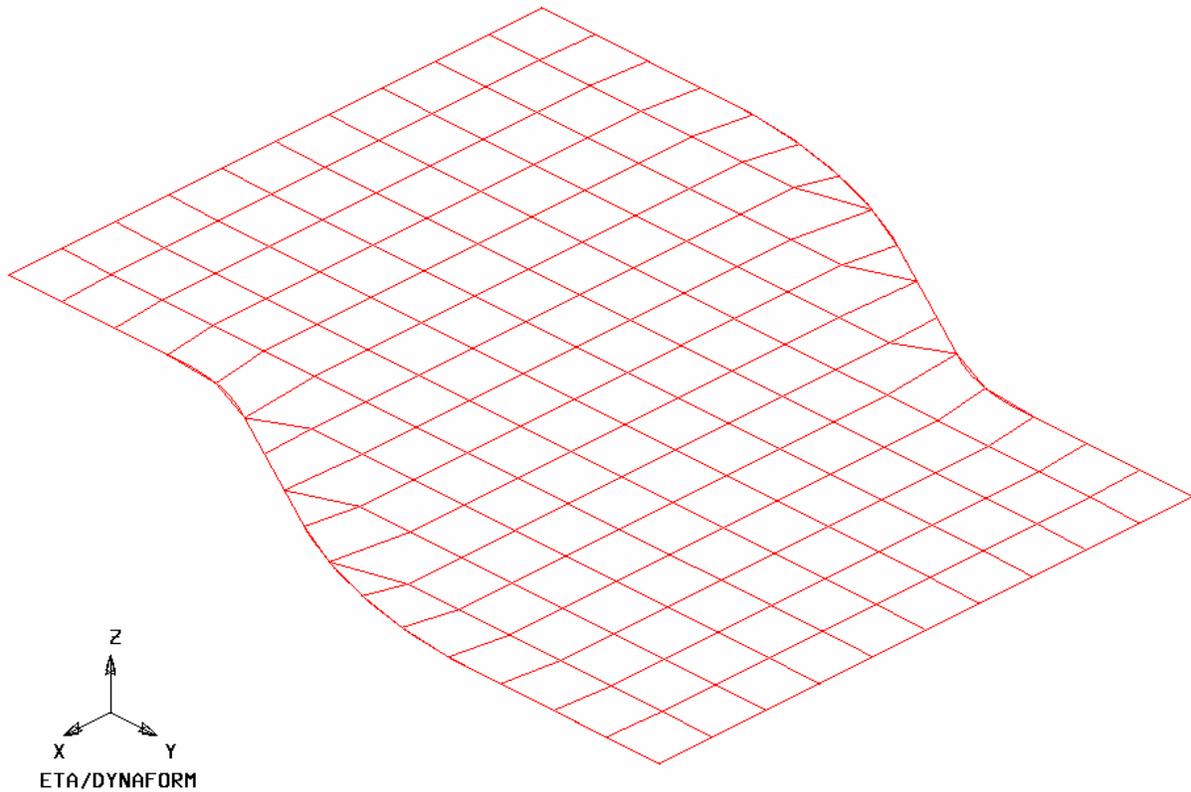


Figure 25: Blank mesh

## II. MESHING SURFACE DATA

Most of the meshing done in eta/DYNAFORM is carried out using the **Surface Mesh** function. This function will automatically create a mesh based on the provided surface data. It provides a very quick and easy way of meshing the tools.

1. Turn off the part **BLANK.LI** and turn on the part **LOWTOOL**. Set the part **LOWTOOL** as current.
2. Select **Preprocess** → **Elements** on **Menu bar**.

<u>L</u> ine/Point	Ctrl+L
<u>S</u> urface	Ctrl+S
<b>E</b> lement	Ctrl+E
<u>N</u> ode	Ctrl+N
<u>M</u> odel Check/Repair	Ctrl+R
<u>B</u> oundary Condition	Ctrl+U
<u>N</u> ode/Element <u>S</u> et	Ctrl+V

Figure 26: Preprocess menu

3. Select **Surface Mesh** from the **Element** dialog box shown in Figure 27.



Figure 27: Element dialog box

4. In the displayed **Surface Mesh** dialog box (shown in Figure 28), default values in all fields will be used. Toggle off the **In Original Part** and **Boundary Check** options.

**Note:** *Chordal deviation* controls the number of elements along the line/surface curvature;  
*Angle* controls the feature line  
*Gap Tol.* controls whether two adjacent surfaces are connected.

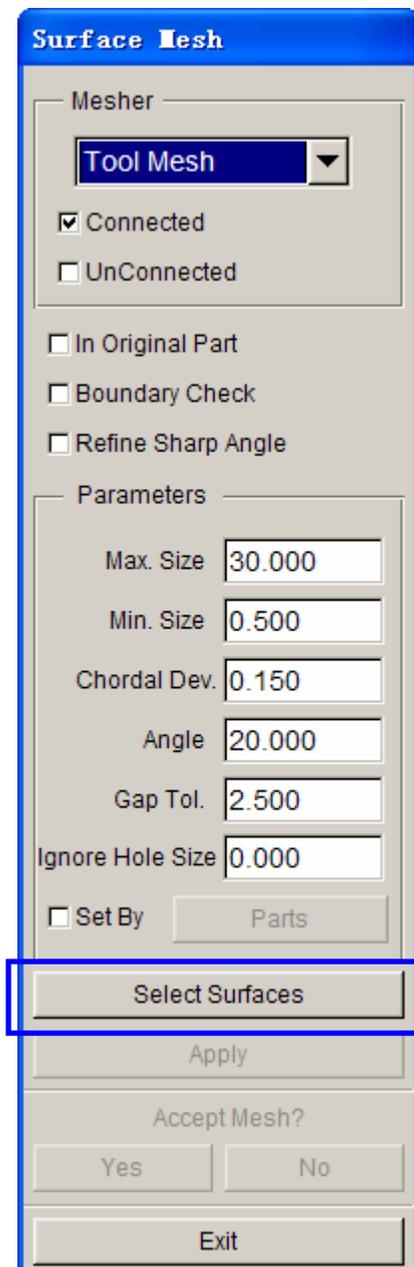


Figure 28: Surface mesh dialog box

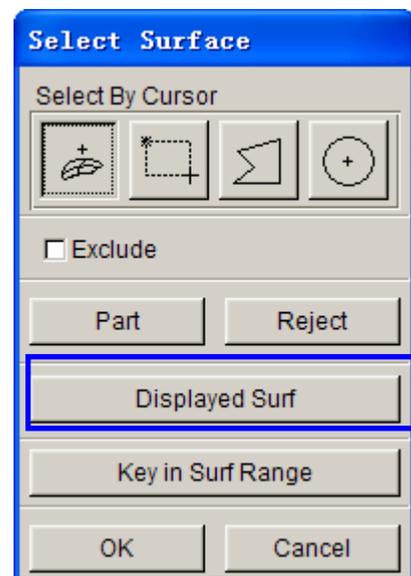


Figure 29: Select surface dialog box

5. Choose the **Select Surfaces** button from **Surface Mesh** dialog.
6. From the **Select Surface** dialog shown in Figure 29, choose the **Displayed Surf** icon.

Notice all of the displayed surfaces will turn white. This verifies they have been selected. The Select Surface dialog box allows you to select the surface(s) in different ways. Place the cursor over each icon and button to identify its function.

- Click **Apply** button on the **Surface Mesh** dialog box.
- The mesh will be created and will be displayed in white. To accept the mesh, click the **Yes** button when prompted, “**Accept Mesh?**” in the **Surface Mesh** dialog box. Compare your mesh with the mesh displayed in Figure 30.

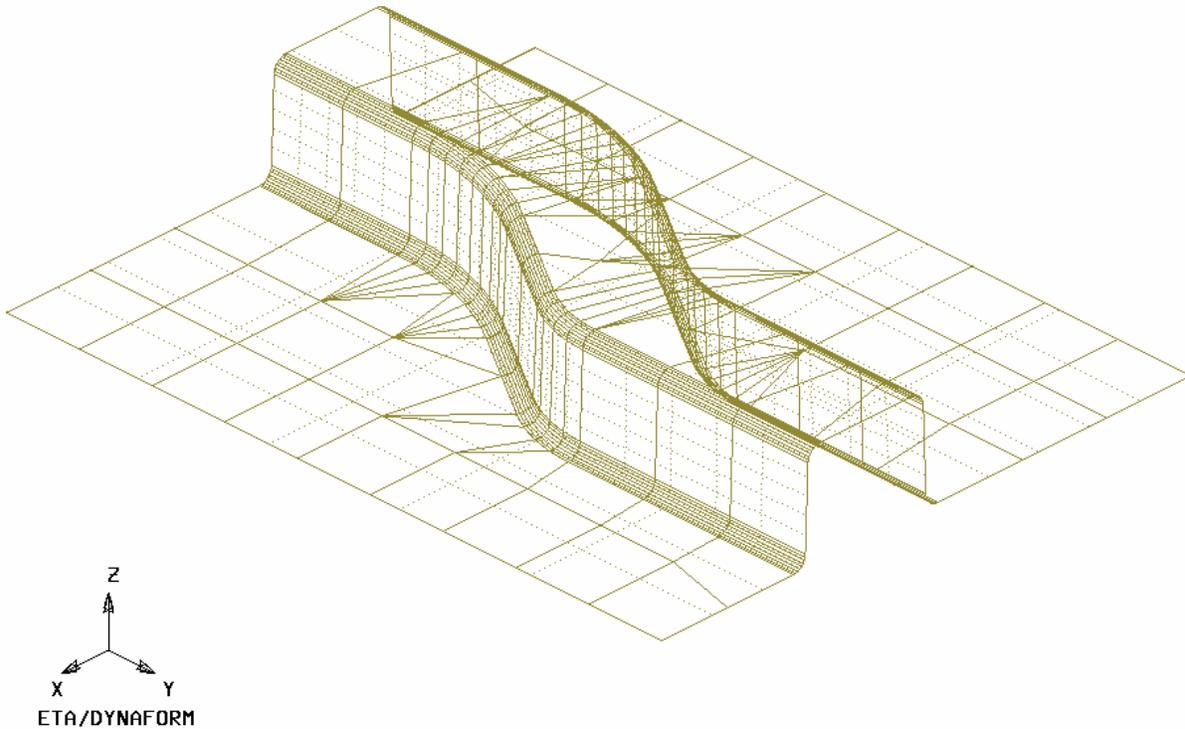


Figure 30: Tool mesh

- Press **Exit** on the **Surface Mesh** dialog box to exit the function.

Now that we have all the parts meshed, you can turn off the surfaces and lines by turning off **Surfaces** and **Lines** in the **Display Options** dialog. This makes it easier to view the mesh.

- Save your database.

### III. MESH CHECKING

As the mesh has been created, its quality has to be checked to verify that there aren't any defects that could cause potential problems in the stamping simulation.

All the utilities used for checking the mesh are located under the **Preprocess**➔**Model Check/Repair** on the **Menu bar**.

<u>L</u> ine/Point	Ctrl+L
<u>S</u> urface	Ctrl+S
<u>E</u> lement	Ctrl+E
<u>N</u> ode	Ctrl+N
<b><u>M</u>odel Check/Repair</b>	<b>Ctrl+R</b>
<u>B</u> oundary Condition	Ctrl+U
<u>N</u> ode/Element <u>S</u> et	Ctrl+V

Figure 31: Preprocess menu

As shown in Figure 31, the **Model Check/Repair** menu consists of several functions that enable you to check the quality of mesh. Only two of the functions are described in this training manual. Please refer to **eta/DYNAFORM Online Help** for information regarding to the remaining functions.

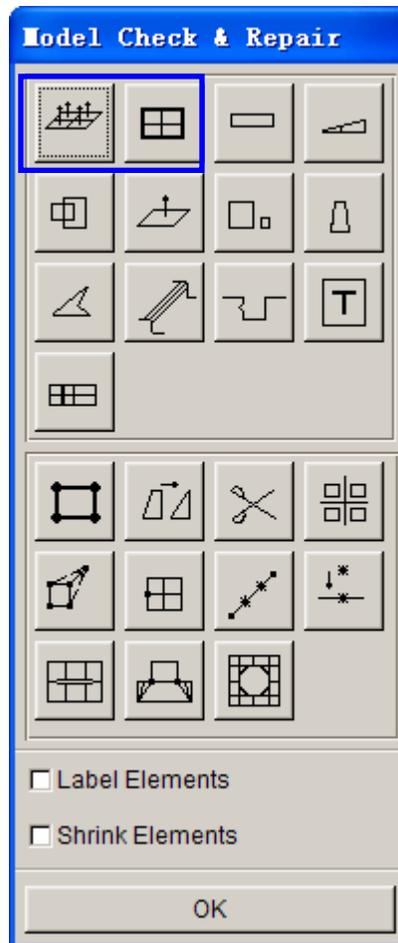


Figure 32: Mesh check &amp; repair dialog box

### Auto Plate Normal



1. Select **Auto Plate Normal** from the **Model Check/Repair** dialog box shown in Figure 32. A new dialog will be displayed.
2. The displayed dialog prompts you to pick an element to check all the active parts or an individual part for element normal consistency. Select an element on the part **LOWTOOL**, as highlighted in Figure 33.

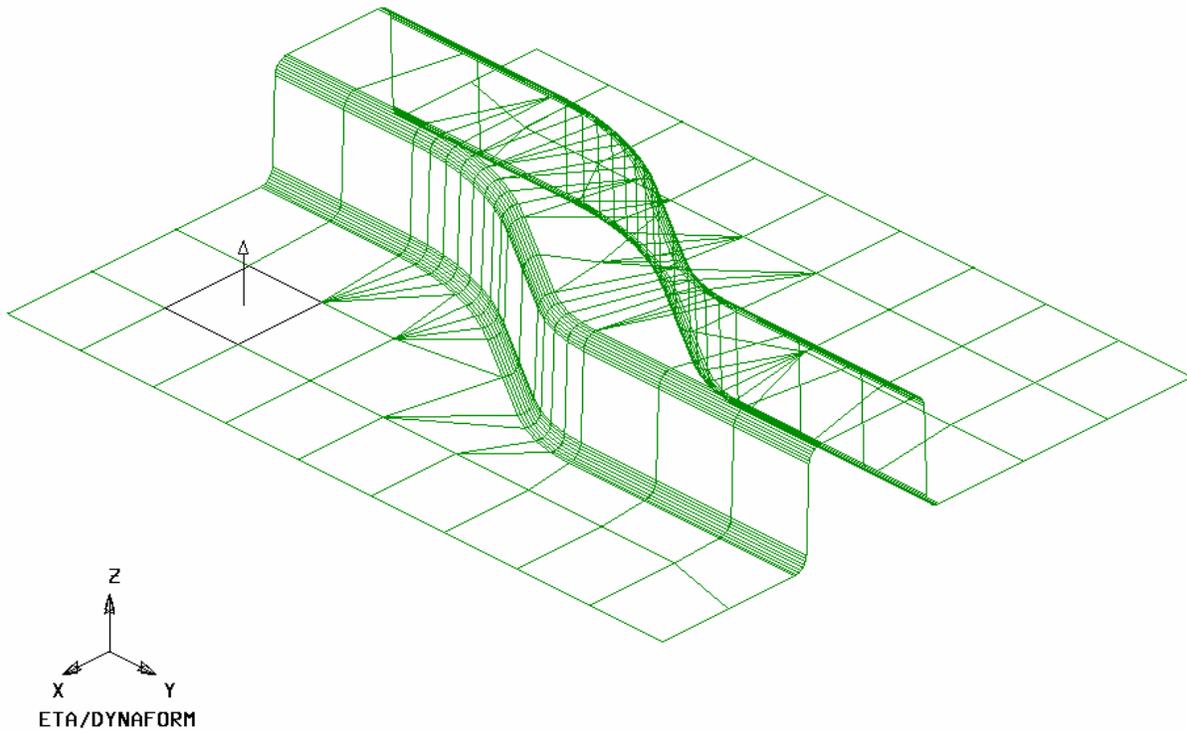


Figure 33: Reference element for normal direction of elements

3. An arrow will be displayed showing the normal direction of the selected element. From the pop up dialog box, click **Yes** to accept the normal direction.

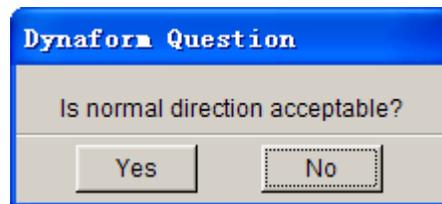


Figure 34: Dynaform Question dialog box

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 direction.

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 direction. The normal direction of all elements in a part has to be consistent. If some elements' normal is pointing upward, while some pointing downward, eta/DYNAFORM will create inferior elements during element offset operation.

**Note:** In this case, we will offset upper tool from the lower tool by the normal direction. Therefore, select **Yes** to make sure the normal to point in the direction of the displayed arrow.

4. Now that the **LOWTOOL** elements are consistent, check the rest of the parts in the database. Turn off all the parts and turn each one at a time. Check the normal direction and make sure it is consistent.
5. Once all the normal directions are consistent, turn on all of the parts and save the changes.

### Display Model Boundary



This function checks any gaps or holes of the part, and highlight them so you can manually correct the problems.

1. Select **Display Model Boundary** from the **Model Check & Repair** dialog box.
2. Minor gaps in the tool mesh are acceptable. Blank mesh should not contain any gaps unless the blank is lanced or is designed with gaps. Select the isometric view and make sure that your display resembles Figure 35.

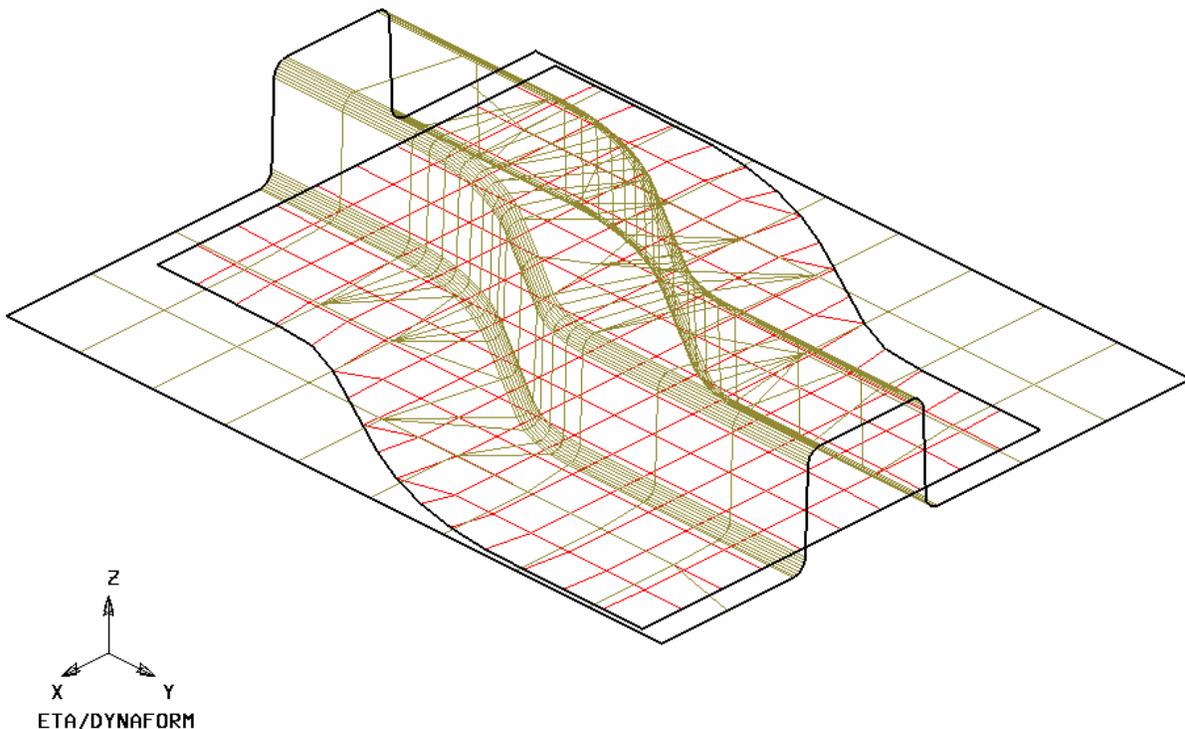


Figure 33: Display model boundary

- Turn off all of the elements and nodes from the **Display Options** dialog (**Note:** the boundary lines are still displayed). This allows you to inspect any small gaps that might be difficult to locate when the mesh is displayed. The results are shown in Figure 36.

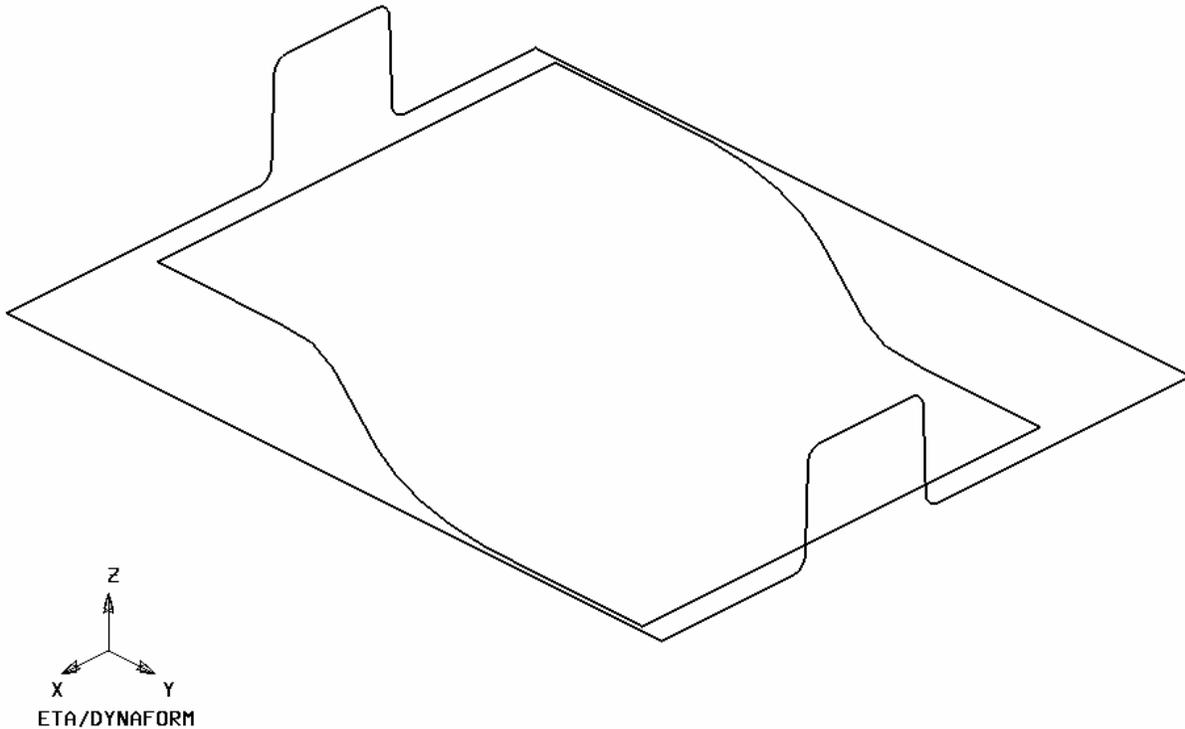


Figure 36: Model boundary after hidden nodes and elements

- Check for overlapping elements and minimum element size. Delete the duplicate elements if there were found.
- Turn only the part **LOWTOOL** on and click the **Clear** icon (as shown in Figure 37) on the **Icon bar** to remove the boundary lines.



Figure 37: Clear icon

- Save your database.

## IV. QUICKSETUP VS. TRADITIONAL SETUP

In eta/DYNAFORM, you can set up the stamping simulation model using either QuickSetup or Traditional setup. Save the changes of your initial database to a unique name, such as

“**original\_mesh.df**”. We will use the database later on to setup stamping simulation model using Traditional setup.

Now, use the **Save As** function to save the database under a different name, for example, “**dftraining\_qs.df**”. We will use this database to proceed with setup of stamping simulation using QuickSetup. The new database name and directory path should be displayed in the upper left corner of the eta/DYNAFORM interface.

This manual will first detail the QuickSetup process, then the Traditional setup. Proceed to the Traditional setup section (page 68) in this manual if you are already familiar with the QuickSetup interface and analysis setup procedure.

# QUICKSETUP

Prior to entering the QuickSetup interface, you need to separate the binder run-out (lower ring) from the lower tool. This will allow the QuickSetup to automatically offset the upper binder from this run-out. This procedure is common to all QuickSetup models that require a binder.

## I. DEFINE THE LOWER RING FROM THE LOWER TOOL

Next step is to separate the **Lower Ring** from the **LOWTOOL**, and move elements on run-out of **LOWTOOL** into **Lower Ring**.

1. Turn on **LOWTOOL** and turn off all other parts.
2. Create a new part called **LOWRING**. This part will keep the elements separated from the **LOWTOOL**. Click **Parts**⇒**Create** on **Menu bar**. See Figure 38.

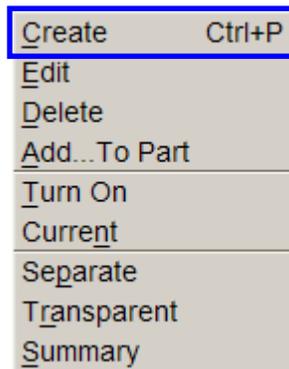


Figure 38: Preprocess menu

3. Enter **LOWRING** at the input field of *Name* in the **Create Part** dialog box illustrated in Figure 39. Click **OK** to create the part.

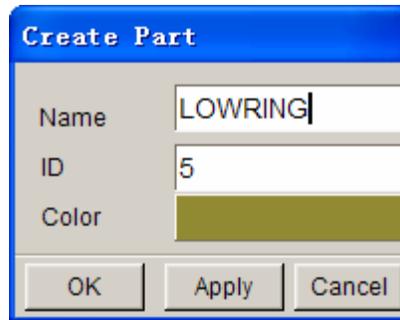


Figure 39: Create part dialog box

4. The part **LOWRING** has been created and set as the current part automatically. You can now place the lower ring elements in this part.
5. Click on **Parts**→**Add... To Part** on **Menu bar**. See Figure 40.

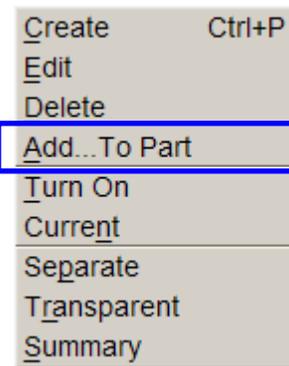


Figure 40: Preprocess menu

6. The **Add...To Part** dialog box shown in Figure 41 is displayed. Click the **Element(s)** button to display the **Select Elements** dialog box shown in Figure 42.

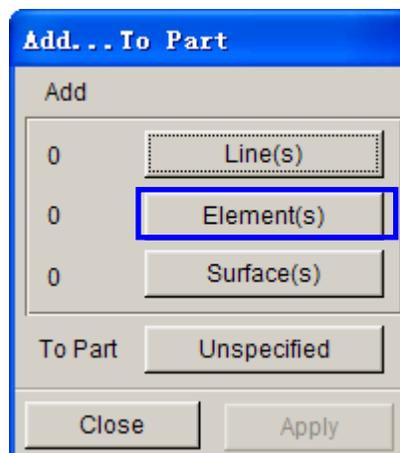


Figure 41: Add...To Part dialog box

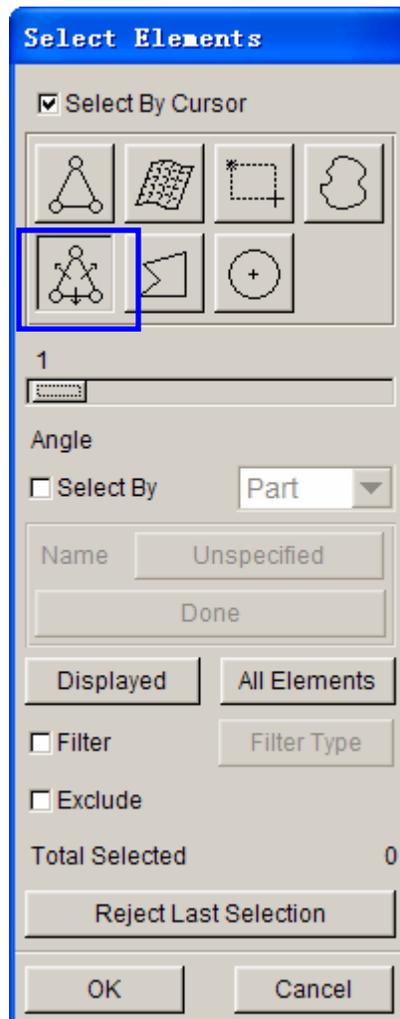


Figure 42: Select elements dialog box

7. The easiest way to select all elements of the ring is to switch the view to the **YX plane** using the **Top View** icon on the **Icon bar**. Then, select the **Spread** icon shown in Figure 42, following by pressing and holding left mouse button on the **Angle Slider**. Move your mouse to drag the slider. Since the ring surface is flat, you may set **1°** as the spread angle.
8. Now, use your mouse cursor to pick the element (as illustrated in Figure 43) on the lower right corner of **LOWTOOL**.

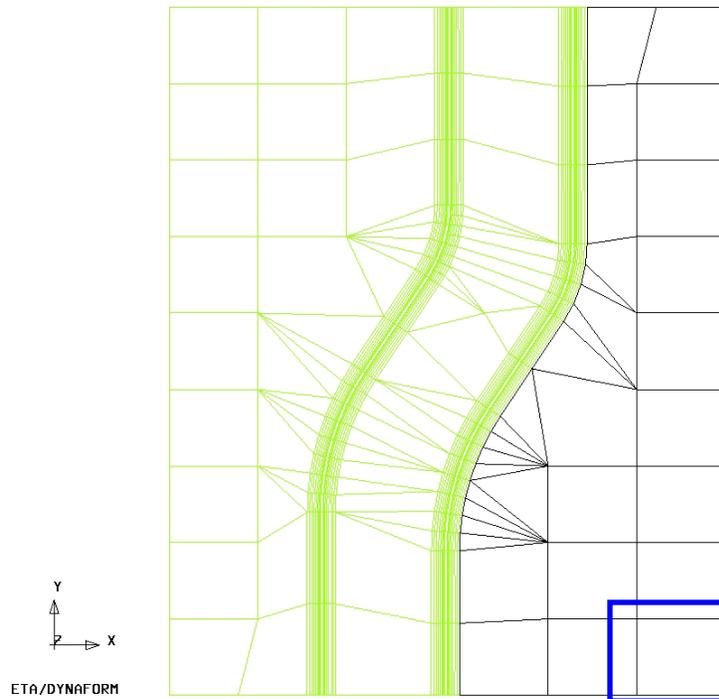


Figure 43: Selecting corner element

All elements in the flat area before an element angle change of larger than  $1^\circ$  should be highlighted. Compare your display to Figure 43. If your results differ, repeat the above steps.

9. Use your mouse cursor to pick the element on lower left corner of **LOWTOOL**.
10. Select **OK** on the **Select Elements** dialog box. You will find the number of elements (67) is shown on the left side of the **Element(s)** button as below, as indicated in Figure 44.
11. Select the **Unspecified** button to display the **Part Turn On/Off** dialog box shown in Figure 45.

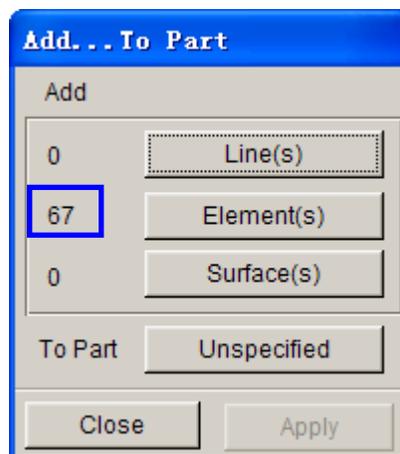


Figure 44: Indicating selected elements

12. Select **LOWRING** in the **Select by Name** list. The program changes the **Unspecified** button to **LOWRING**.

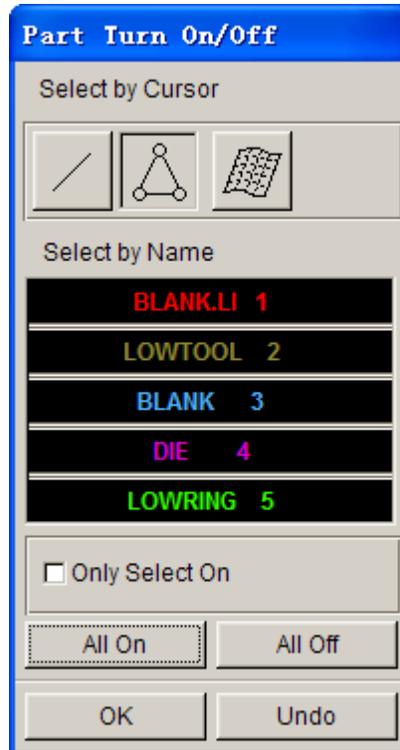


Figure 45: Part turn on/off dialog box

13. The **Apply** button is enabled, as shown in Figure 46. Click **Apply** button to move all selected elements into **LOWRING**.

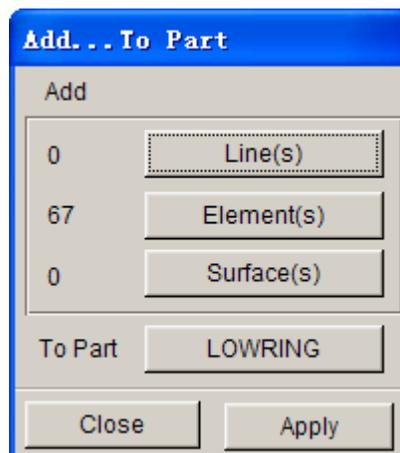


Figure 46: Add...To Part dialog box

- Hit the **Top View** icon in the Icon bar. Then, use the **Part Turn On/Off** function to display the **LOWRING** illustrated in Figure 47. If display on your screen differs, repeat the above steps.

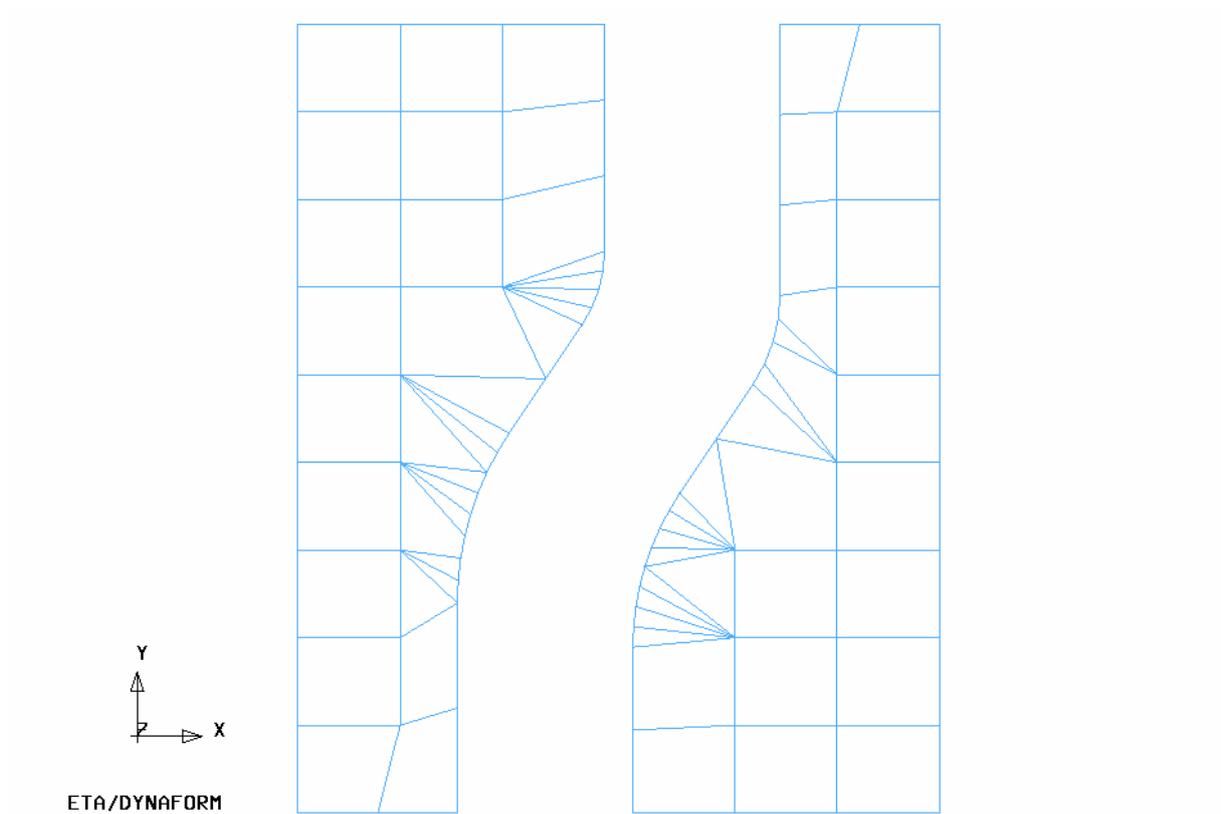


Figure 47: Illustration of binder surface

- Save your database.

## II. QUICKSETUP INTERFACE

- Select **Setup** → **Draw Die** illustrated in Figure 48 to display the QuickSetup GUI.

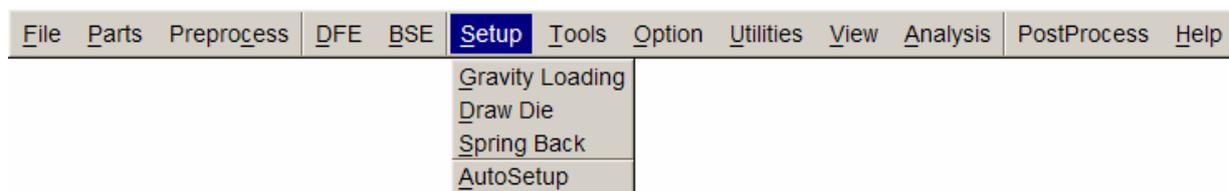


Figure 48: Setup menu

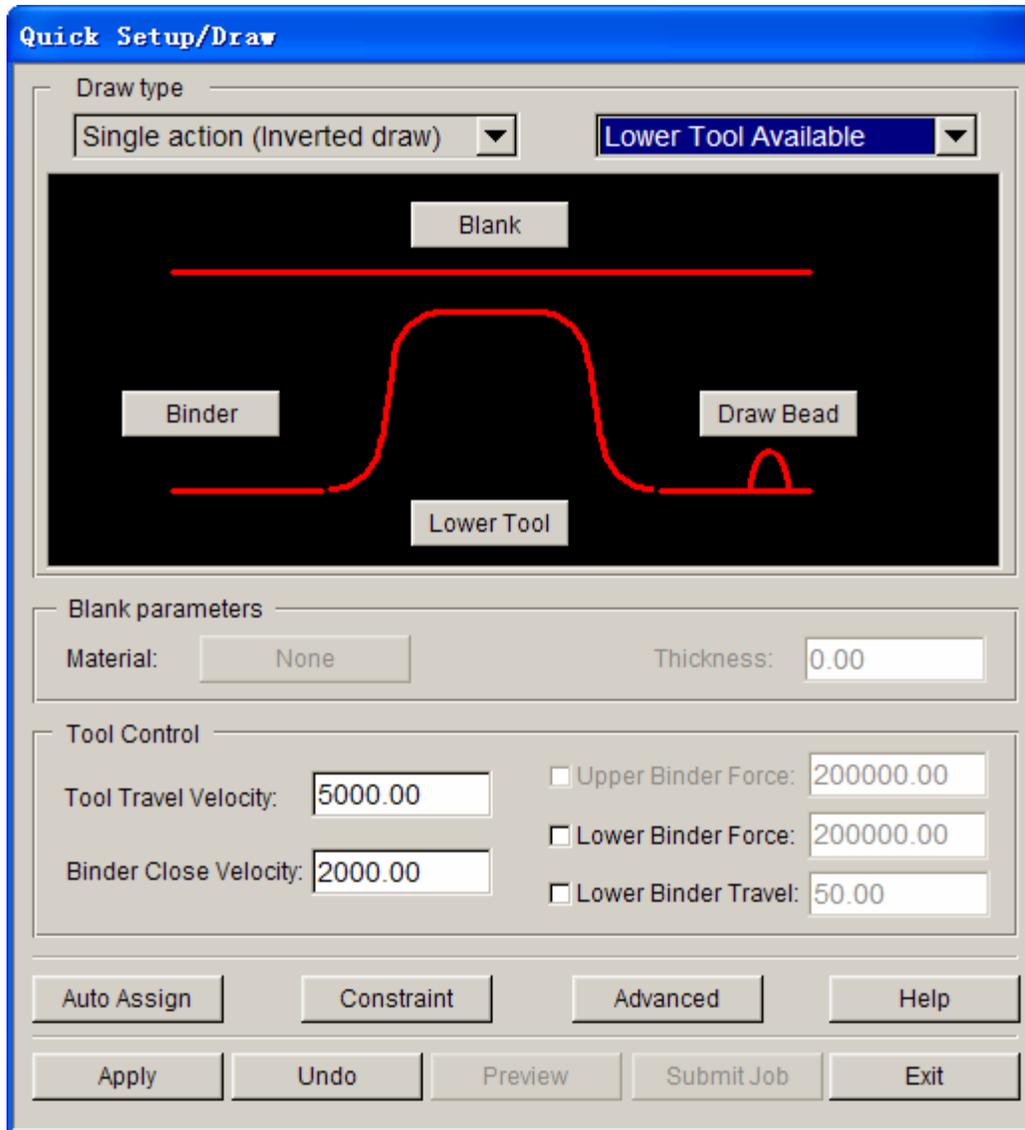


Figure 49: QuickSetup GUI

2. As shown in Figure 49, the undefined tools are highlighted in red in the QuickSetup GUI. Firstly, you need to select the **Draw Type** and **Type of Available tool**. In this application, the draw type is “**Single action**” or **Inverted draw**, while the **lower tool** is selected as the available tool.
3. Define the blank and tools by clicking the corresponding buttons.

### III. DEFINE TOOLS

The parts **LOWTOOL** and **LOWRING** are meshed and can be defined as the Binder and Lower Tool, respectively.

To define the Binder:

1. Click the **Binder** button, following by selecting the **SELECT PART** function from the **Define Tool** dialog box illustrated in Figure 50.
2. Next, click on the **Add** button in the **Define Binder** dialog box shown in Figure 51.
3. Use your mouse cursor to pick the part **LOWRING** from the **Select Part** dialog box illustrated in Figure 52.
4. Click **OK** button in the **Select Part** dialog box to confirm part selection.
5. Again, click **OK** button to dismiss the **Define Binder** dialog box.
6. Click **EXIT** button to quit the **Define Tool** dialog box.
7. Repeat the above steps to define the **Lower Tool**. Once both tools are defined, the color in the Quick Setup/Draw window will be changed to green, as illustrated in Figure 53.



Figure 50: Define tool dialog box

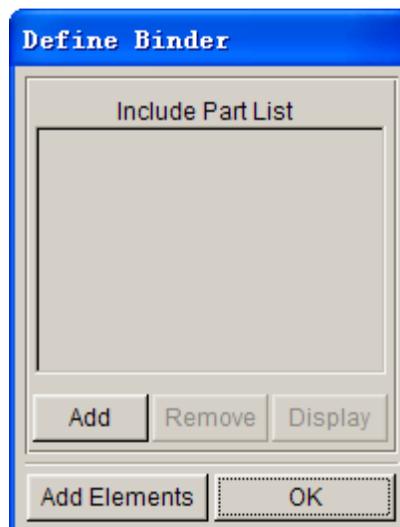


Figure 51: Define binder dialog box

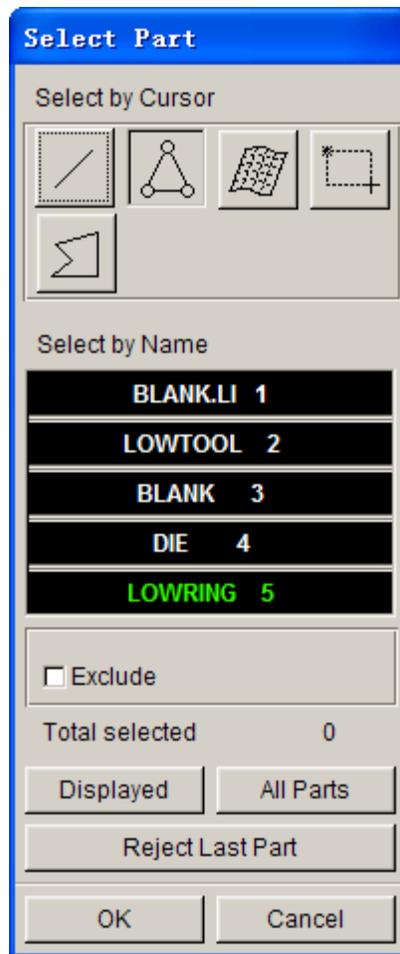


Figure 52: Select part dialog box

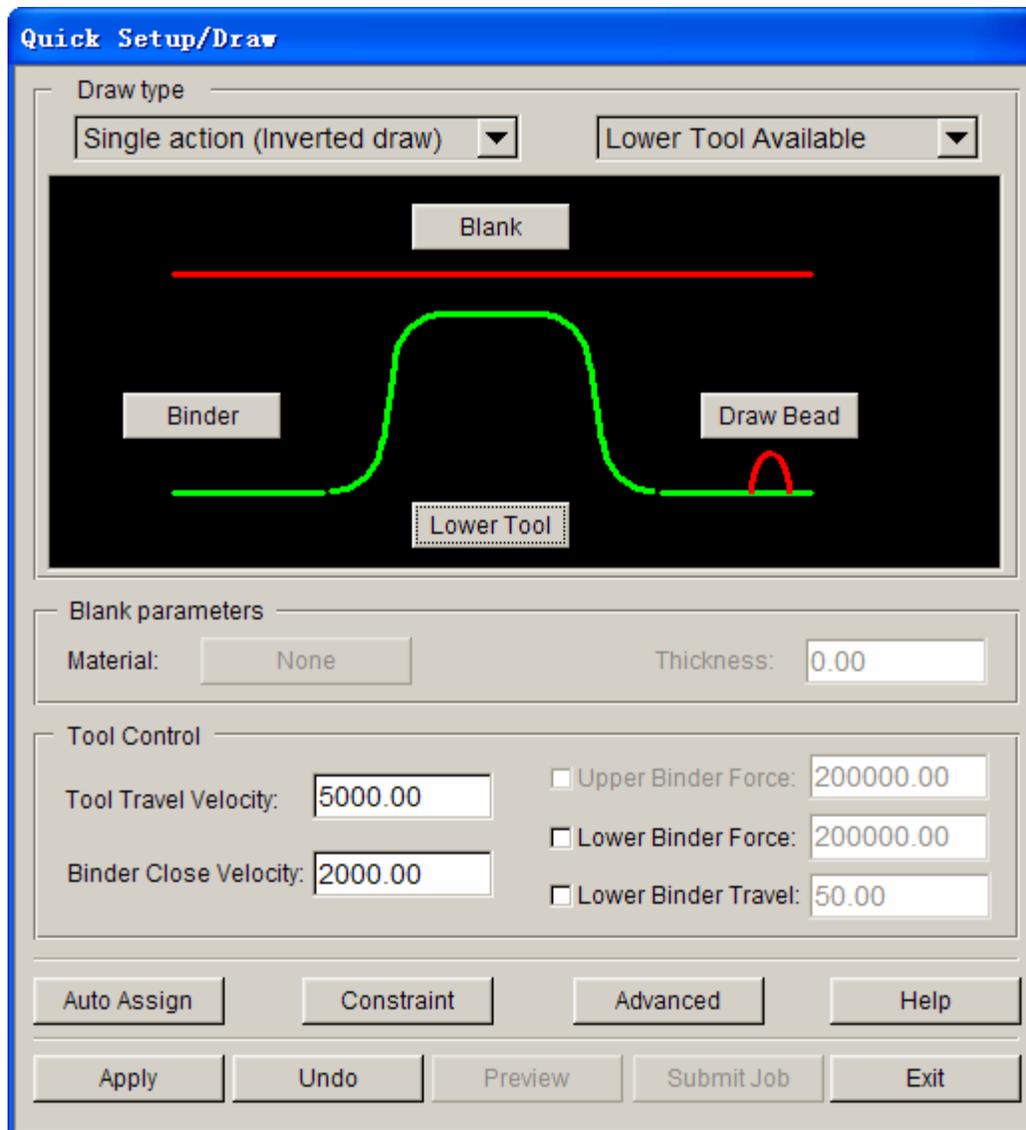


Figure 53: QuickSetup GUI after tools are defined

#### IV. DEFINING THE BLANK MATERIAL

1. Click the **Blank** button from QuickSetup GUI.
2. Then, select the **SELECT PART** function from the **DEFINE BLANK** dialog box.
3. Click “**Add**” button to pick the **BLANK** from the part list in **Select Part** dialog box.
4. Click **OK** button in the **Select Part** dialog box to confirm part selection.
5. Click **EXIT** button to quit the **DEFINE BLANK** dialog box.

6. Next, you need to define the material and thickness. You may enter the blank thickness value in the input data field of **Thickness**. In this case, you select the default value of **1.0** (mm).
7. From the **Blank Parameters** field, click None button to assign the blank material. From the Material dialog box, click the **Material Library** button to display the generic material library illustrated in Figure 54.

Select the **Mild Steel “DQSK”** under material type 36 (**Note:** material type 36 and 37 are recommended for most simulations).

Dynaform Material Library		Type 1	Type 18	Type 24	Type 36	Type 37	Type 39	Type 64
Strength Level	Material Name	ELASTIC	POWER	LINEAR	3-PARAM	ANISOTR	FLD_TRA	RATE_SEN
STEEL	Mild	DQ	+	+	+	+	-	-
		DQSK	+	+	+	+	-	-
		DDQIF	+	+	+	+	-	-
		CQ	-	+	+	+	-	-
	Medium	BH180	+	+	+	+	+	-
		BH210	+	+	+	+	+	-
		BH250	+	+	-	+	+	-
		BH280	+	+	+	+	+	-
	High	HSLA250	+	+	+	+	+	-
		HSLA300	+	+	+	+	+	-
		HSLA350	+	+	+	+	+	+
		HSLA420	+	+	+	+	+	-
	Advanced High	DP500	+	+	+	+	+	-
		DP600	+	+	+	+	+	-
	Hot Rolled	CQ	+	+	+	+	+	-
		DQSK	+	+	+	+	+	-
		DDQIF	+	+	+	+	+	-
		HSLA400	+	+	+	+	+	-
	Stainless	SS11CrCb	+	+	+	+	+	-
		SS18CrCb	+	+	+	+	+	-
SS304		+	+	+	+	+	-	
SS409Ni		+	+	+	+	+	-	
ALUMINUM	AA5182	+	+	+	+	+	-	
	AA5454	+	+	+	+	+	-	
	AA5754	+	+	+	+	+	-	
	AA6009	+	+	+	+	+	-	

Figure 54: Generic material library window

8. Select **OK** to use the default material parameters (illustrated in Figure 55) for the DQSK material model (**Disclaimer:** ETA does not guarantee validity or accuracy of the generic material models in the material library. You should contact the original material suppliers to determine appropriate material parameters). To complete the material definition, select **OK** from the material window and return to the QuickSetup interface.
9. Save your database.

Property	Value
MATERIAL TITLE	DQSK
MASS DENSITY	7.850000E-009
YOUNGS MODULUS	2.070000E+005
POISSONS RATIO	2.800000E-001
HARDENING RULE(EXPON.)	3.000000E+000
MATERIAL PARAM P1 (K)	5.204000E+002
MATERIAL PARAM P2 (N)	2.320000E-001
EXPONENT FACE M	6.000000E+000
LANKFORD PARAM R00	1.730000E+000
LANKFORD PARAM R45	1.350000E+000
LANKFORD PARAM R90	2.180000E+000
FLD CURVE NUMBER	0

Figure 55: Material type 36 window

The following are the description of other functions in the QuickSetup interface:

- **Auto Assign** enables you to quickly assign parts with conventional names as tooling. For example, if the blank part name is named “BLANK”, the part “BLANK” will be defined as the model BLANK after you click the **Auto Assign** button. “DIE” and “BINDER” are the other tooling names recognized by **Auto Assign** function in QuickSetup interface. Drawbeads are not recognized by the **Auto Assign** function.
- **Constraint** allows you to define SPC (single point constraint) for symmetry or other boundary conditions.
- **Advanced** allows you to change default parameters related to QuickSetup.
- **Apply** automatically copy all defined tooling with its mating tool, in addition to assigning the travel curves.
- **Reset** deletes all mating tools and travel curves “resetting” the database to the state it was before the apply button was selected.
- **Preview** allows you to animate tool movement and check the tooling travel curves.
- **Submit job** initiates the analysis menu.
- **Exit** is utilized to exit the QuickSetup interface.

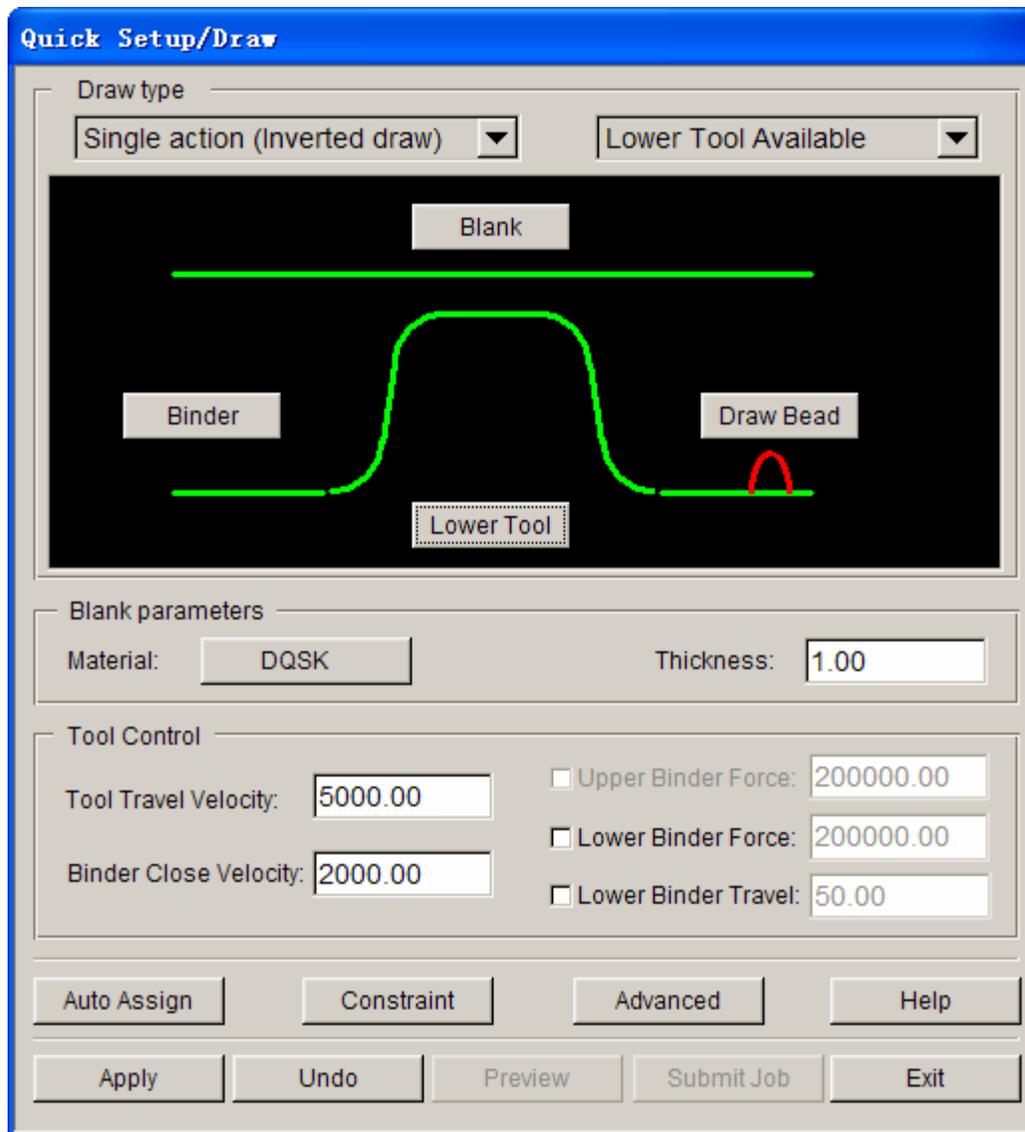


Figure 56: QuickSetup GUI after tools and blank are defined

10. Now, select **Apply** illustrated in Figure 56. The QuickSetup will automatically create mating tools, position the tools and generate the corresponding travel/force curves. Both **Preview** and **Submit Job** function are activated.
11. Select **Preview** to check the tooling movement.
12. Compare your display with the illustration in Figure 57.

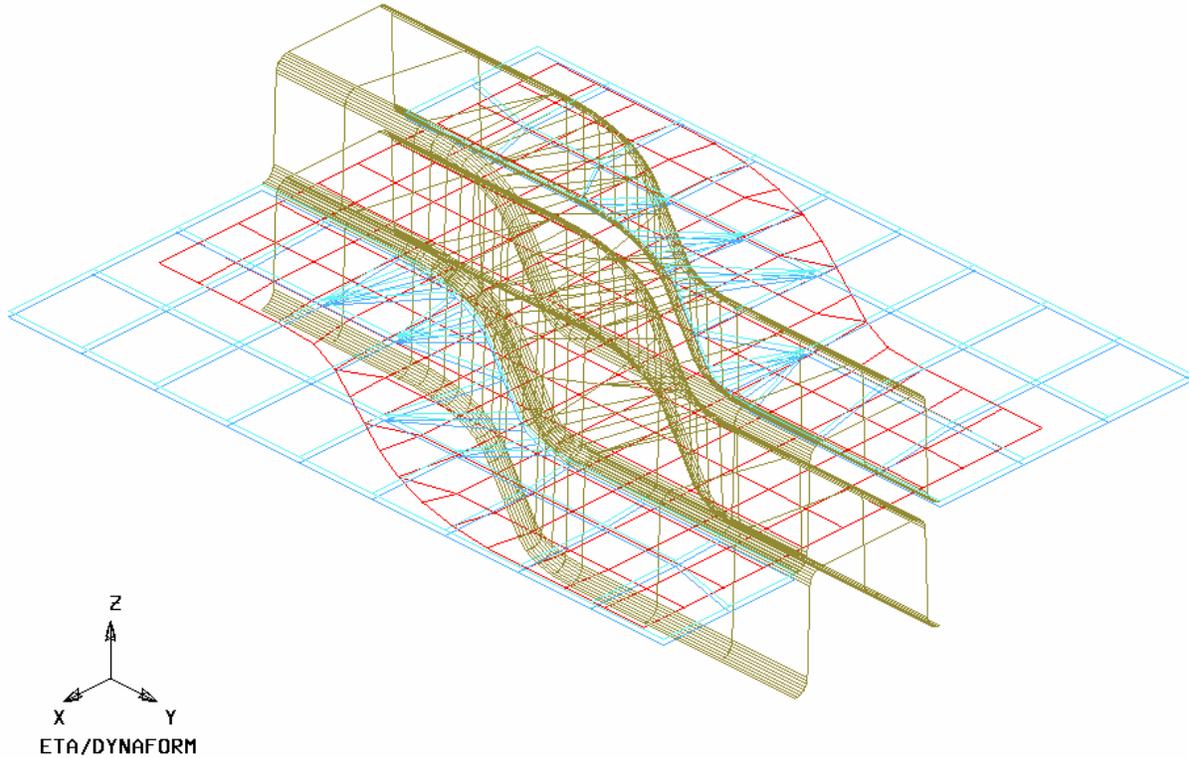


Figure 57: Illustration of model setup after autopositioning

## V. RUNNING THE ANALYSIS

After verifying the tool motion, you can define the final control parameters and proceed to run the analysis.

1. Click **Submit Job** button to display the “**Analysis**” dialog box shown in Figure 58.

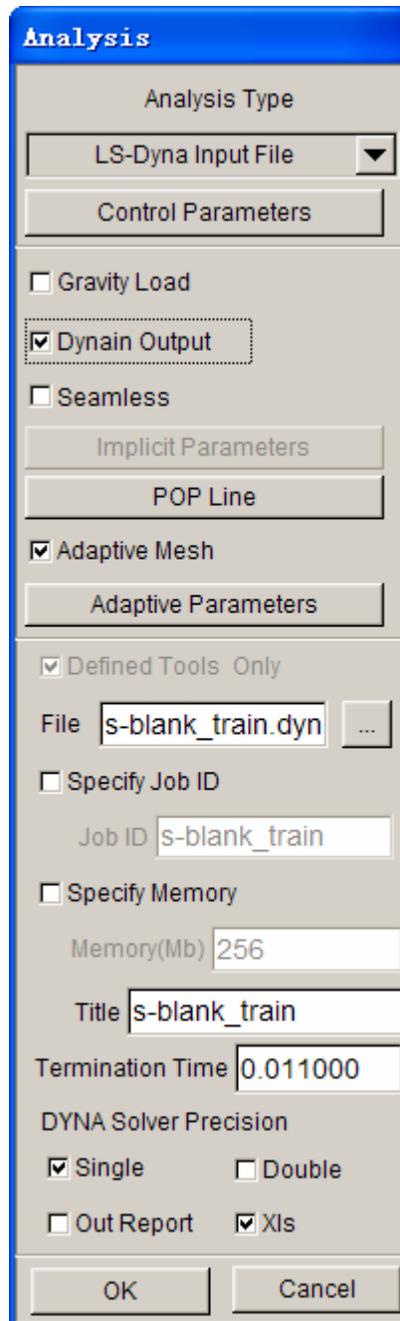


Figure 58: Analysis dialog box

2. Click the **Control Parameters** button in the **Analysis** dialog box to display the DYNA3D CONTROL PARAMETERS dialog box illustrated in Figure 59.

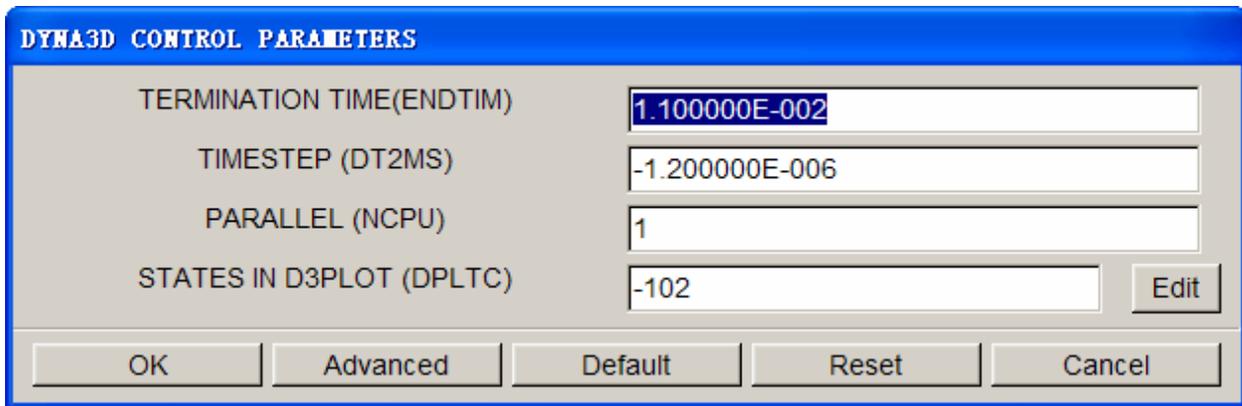


Figure 59: DYNA3D control parameters dialog box

You are recommended to accept the default control parameters. For detailed information about each control parameters, please refer to the *LS-DYNA User's Manual*.

4. Click **OK** button to accept the default control parameters.
5. By default, the **Adaptive Mesh** option is checked. Adaptive mesh allows you to refine the coarse elements into smaller elements to capture detailed tool geometry.
6. Click **Adaptive Parameters** button to display the **ADAPTIVE CONTROL PARAMETERS** dialog box illustrated in Figure 60. Set the **LEVEL (MAXLVL)** as **3**. This means that the mesh will be split twice if needed. Higher levels of adaptivity will result in better accuracy but require longer processing time. Since this is a simple part, adaptive level 3 will be sufficient. The default values will be used for other parameters. Click **OK** button to accept the default setting.

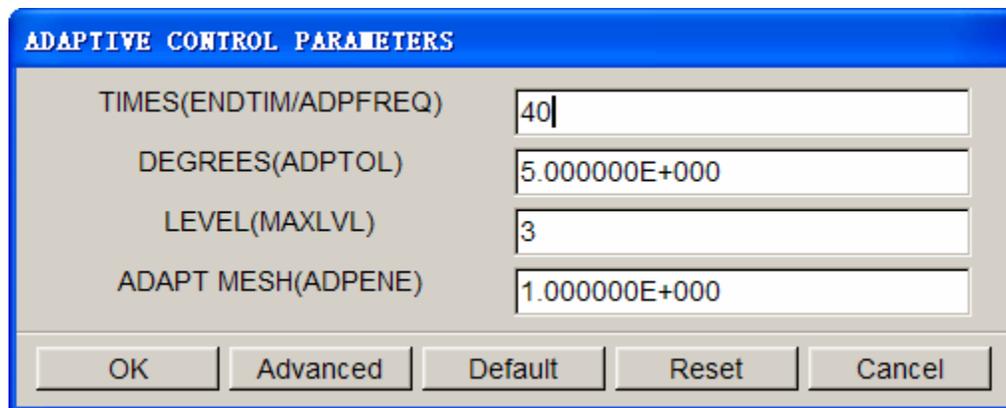


Figure 60: Adaptive control parameters dialog box

7. To submit the job, click the drop down button of **Analysis Type** to select **Full Run Dyna** (as shown in Figure 61). Next, toggle on “**Specify Memory**” checkbox and key in **120** (Mb). Then, click **OK** button to initiate the solver.

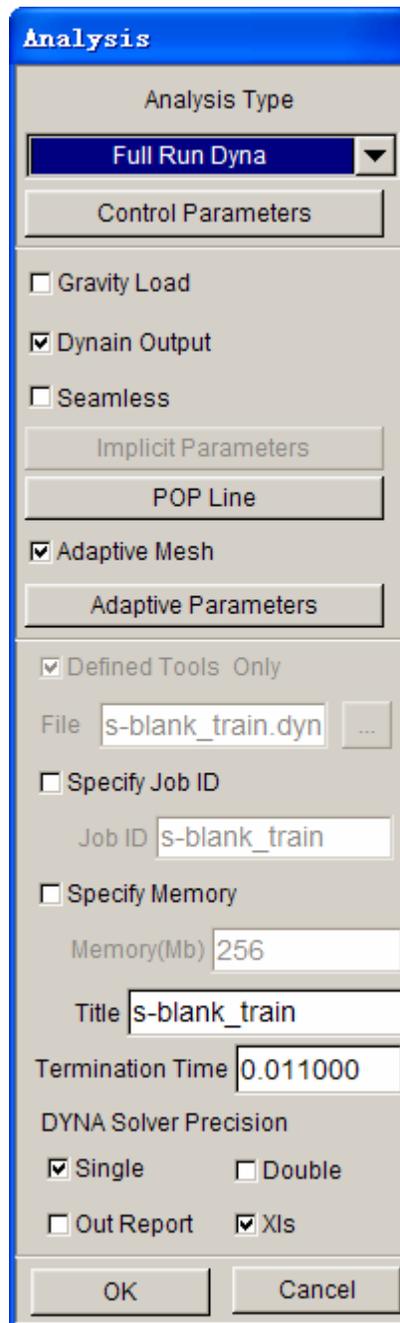


Figure 61: Analysis dialog box

The solver displays a DOS window (illustrated in Figure 62) showing the status of the in-progress simulation. You notice an estimated completion time is given. This time is not accurate due to implementation of adaptivity option. However, it does give you a general idea.

Proceed to page 97 for post-processing after the program is completed.

```

c:\ D:\eta\SOLVER~1\lsdyna.exe
eroded kinetic energy..... 0.00000E+00
eroded internal energy..... 0.00000E+00
total energy..... 1.00000E-20
total energy / initial energy.. 1.00000E+00
energy ratio w/o eroded energy. 1.00000E+00
global x velocity..... 0.00000E+00
global y velocity..... 0.00000E+00
global z velocity..... 0.00000E+00

number of shell elements that
reached the minimum time step.. 0
cpu time per zone cycle..... 0 nanoseconds
average cpu time per zone cycle... 0 nanoseconds
average clock time per zone cycle.. 174593 nanoseconds

estimated total cpu time      =      0 sec <      0 hrs  0 mins>
estimated cpu time to complete =      0 sec <      0 hrs  0 mins>
estimated total clock time    =    2130 sec <      0 hrs 35 mins>
estimated clock time to complete =    2130 sec <      0 hrs 35 mins>

added mass      = 0.0000E+00
percentage increase = 0.0000E+00

1 t 0.0000E+00 dt 1.55E-06 flush i/o buffers
1 t 0.0000E+00 dt 1.55E-06 write d3plot file

```

Figure 62: LS-DYNA solver window

The status of in-progress stamping simulation can be determined by using the sense switch. The user can press the **Ctrl + C** key to interrupt the in-progress simulation. This operation will momentarily pause the solver and prompt the user to input the sense switch code at **“.enter sense switch:”** line. There are nine terminal sense switch codes provided in LS-DYNA. The common sense switch codes are listed below:

- sw1** – A restart file is written and solver terminates.
- sw2** – Solver responds with estimated time and cycle numbers.
- sw3** – Creates a d3dump restart file and solver continues
- sw4** – Creates a d3plot file and solver continues.

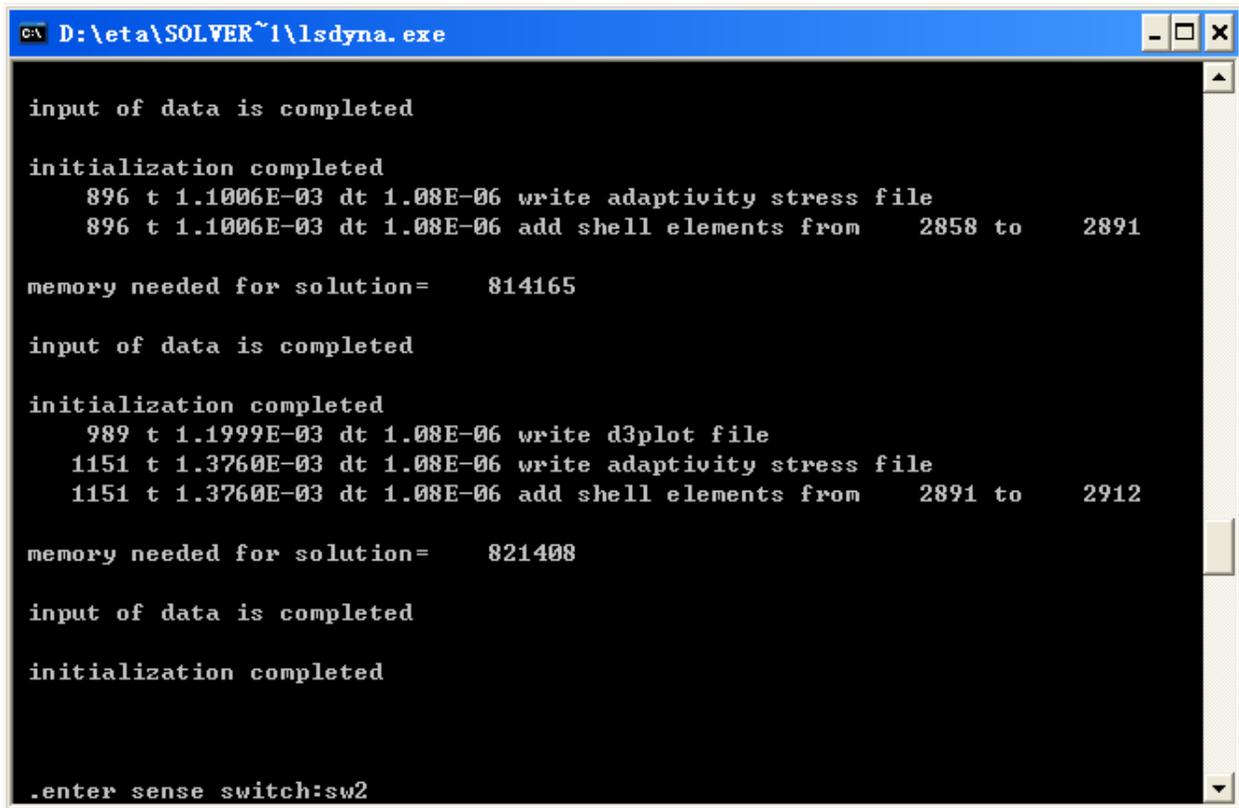
*Note: These switches are case sensitive and must be all lower case when entered. For information about other sense switch codes, refer to LS-DYNA User’s Manual.*

Type in **sw2** and press **Enter**. Notice the estimated time has changed. You can use these switches anytime while the solver is running.

When you submit a job from eta/DYNAFORM, an input deck is automatically created. The input deck is adopted by the solver, LS-DYNA, to process the stamping simulation. The default input

deck names are **yourdatabasename.dyn** and **yourdatabasename.mod**. In addition, an index file named **yourdatabasename.idx** is generated for the reference in eta/POST. The **.dyn** file contains all of the keyword control cards, while the **.mod** file contains the geometry data and boundary conditions. Advanced users are encouraged to study the **.dyn** input file. For more information, refer to the *LS-DYNA User's Manual*.

*Note: All files generated by either eta/DYNAFORM or LS-DYNA is stored in the directory in which the eta/DYNAFORM database has been saved. You shall ensure the folder doesn't include similar files prior to running the current job because these files will be overwritten.*



```
D:\eta\SOLVER~1\lsdyna.exe

input of data is completed

initialization completed
 896 t 1.1006E-03 dt 1.08E-06 write adaptivity stress file
 896 t 1.1006E-03 dt 1.08E-06 add shell elements from 2858 to 2891

memory needed for solution= 814165

input of data is completed

initialization completed
 989 t 1.1999E-03 dt 1.08E-06 write d3plot file
1151 t 1.3760E-03 dt 1.08E-06 write adaptivity stress file
1151 t 1.3760E-03 dt 1.08E-06 add shell elements from 2891 to 2912

memory needed for solution= 821408

input of data is completed

initialization completed

.enter sense switch:sw2
```

Figure 63: Sense switch

Again, the eta/DYNAFORM Quick Setup interface is designed to help you to quickly setup a standard drawing simulation. You are encouraged to learn the traditional setup described in the following section.

# AUTO SETUP

Open the saved database as described in meshing section of this manual (on page 23) **dftraining\_as.df**. This file should contain the clean mesh data of the LOWTOOL layer . If you do not have this database, repeat the procedure in **MESHING** section.

Prior to entering the **AUTOSETUP** interface, you only need mesh tools. Other operations are carried out in **AUTOSETUP** interface, such as **PHYSICAL OFFSET** of element, selection of **CONTACT OFFSET**. You can select **AUTOSETUP** option in the **SETUP** menu (shown in Figure 64) to display the interface.

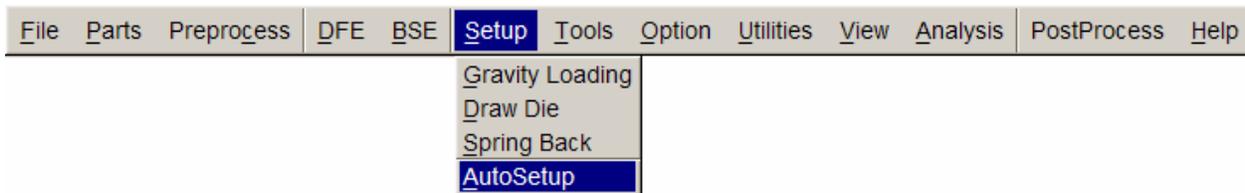


Figure 64: Setup menu

## I. NEW SIMULATION

After you select the **AutoSetup** option, the New Simulation dialog box illustrated in Figure 65 is displayed.

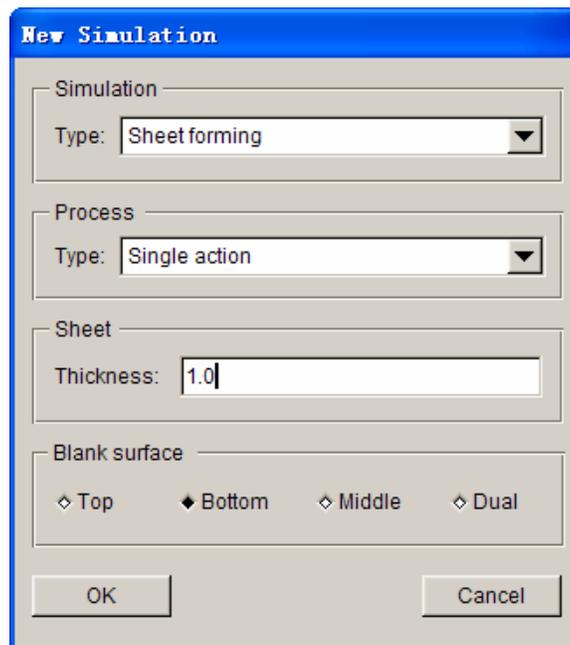
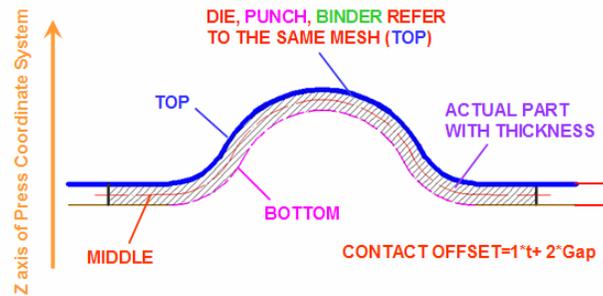


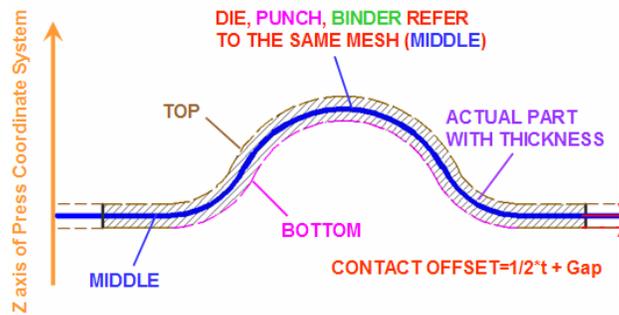
Figure 65: New simulation dialog box

- a) Select simulation type: **Sheet Forming** .

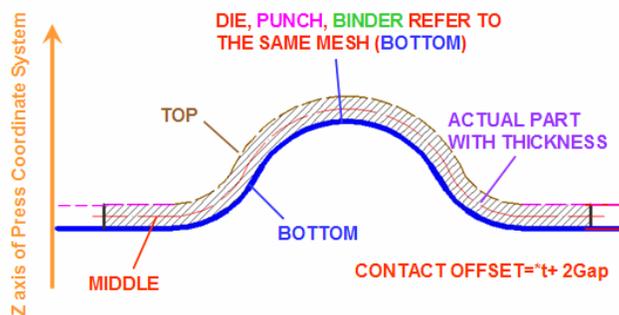
- b) Select process type: **Single Action**.
- c) Input blank thickness: **1.0** (mm).
- d) Select tools reference: **BOTTOM**.
- e) Click **OK** to display the main **AUTOSETUP** dialog box.



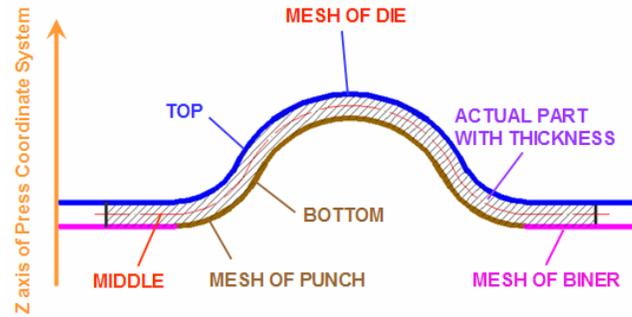
Top surface of part



Middle surface of part



Bottom surface of part



Different surface referred by Die and Punch

Figure 66: Reference blank surfaces

## II. GENERAL

After entering **General** definition page, modify the **Title** to **Df\_training\_01** (as illustrated in Figure 67), while keeping other default parameters.

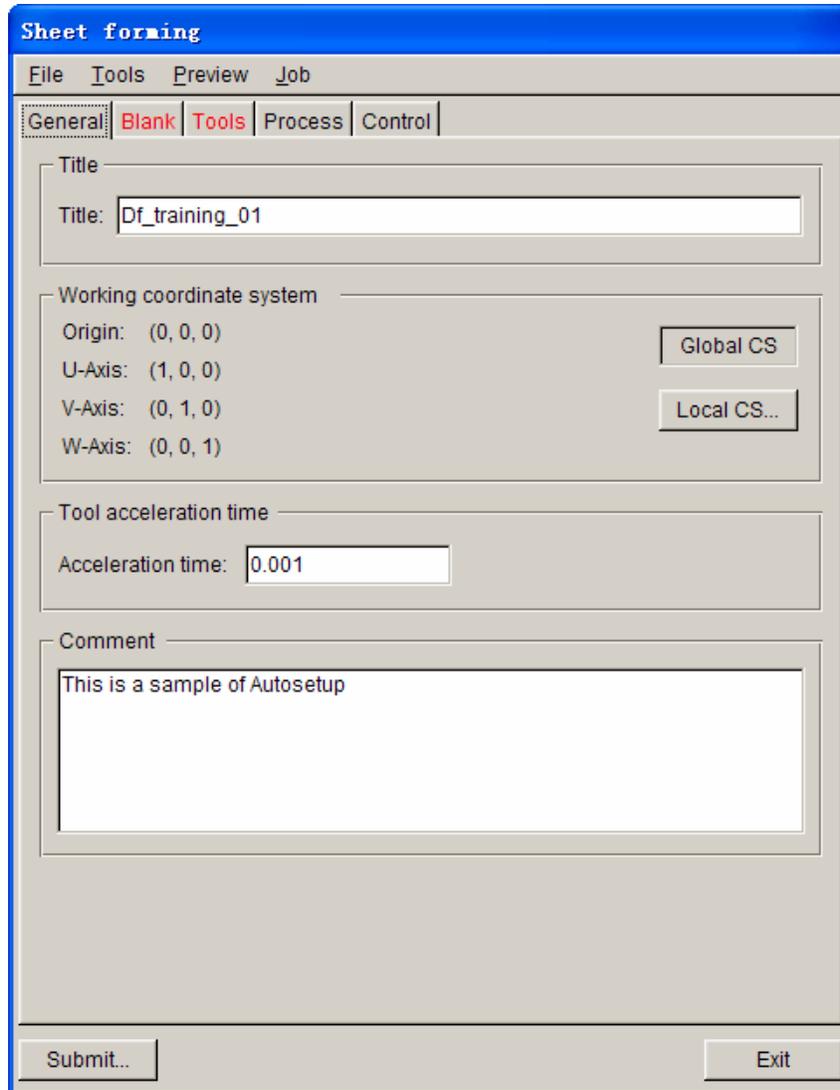


Figure 67: General definition page

### III. BLANK DEFINITION

1. Click the **Blank** tab to enter the **Blank** definition page.
2. Click **Parts...** button from **Geometry** field in **Blank** definition page (shown in Figure 68).

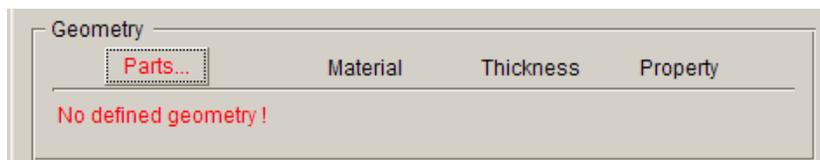


Figure 68: Geometry field

3. The program will pop up the **Define Geometry** dialog box shown in Figure 69.

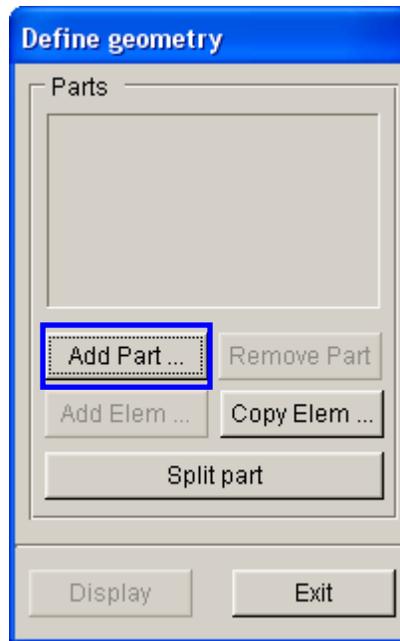


Figure 69: Define geometry dialog box

4. Click **Add Part...** button to display the **Select Part** dialog box. Use your mouse cursor to pick the **BLANK** part in the dialog box (shown in Figure 70).

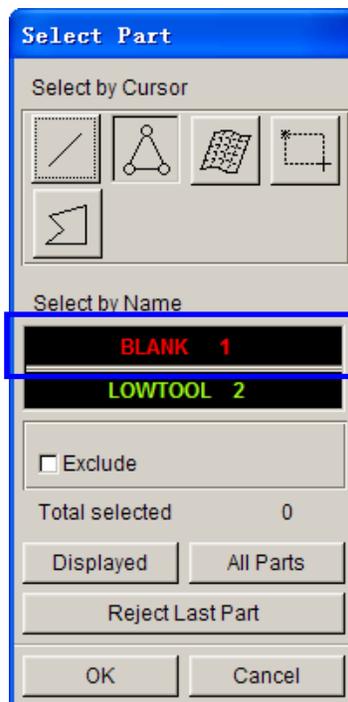


Figure 70: Select Part dialog box

5. After selecting the part, click **OK** button to exit the dialog box. The program returns the **Define Geometry** dialog box. The **BLANK** part is added to the list of Define Geometry

dialog illustrated in Figure 71.

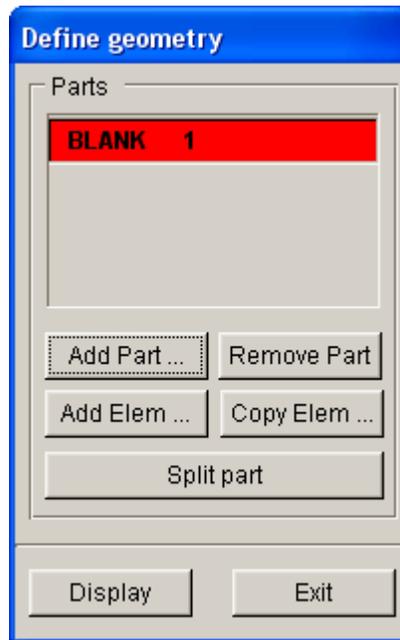


Figure 71: Define geometry dialog box

6. Exit the dialog box to return to the **Blank** definition page. Now, the tag color of the **Blank** is changed red into black, indicating the blank part is assigned. See Figure 72.

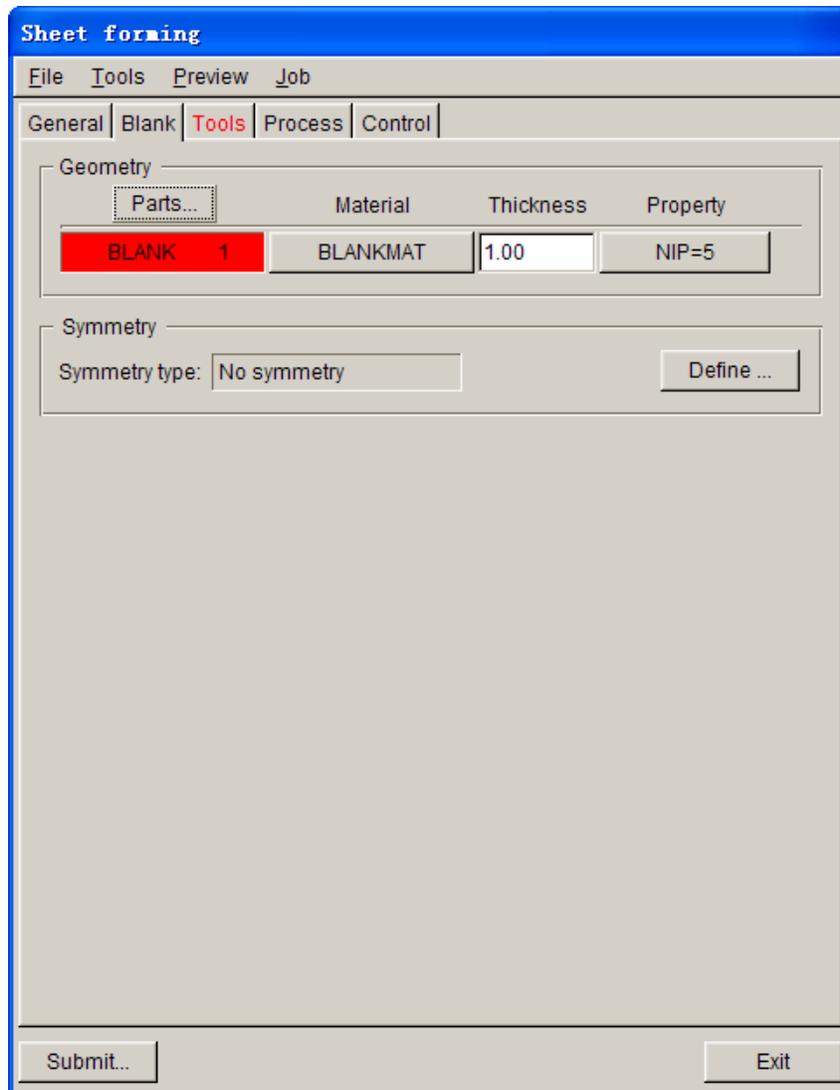


Figure 72: Blank definition page

## IV. TOOLS DEFINITION

1. Click the **Tools** tab to enter the **Tools** definition page.
2. Click  icon in the **Icon bar** to display the **Part Turn On/Off dialog box**, following by turning off the **BLANK** part.
3. **AUTOSETUP** displays three standard tools: die, punch and binder located at the left of **Tools** definition page.
4. From **die** interface illustrated in Figure 73, select **Parts...** button to assign a part as die.

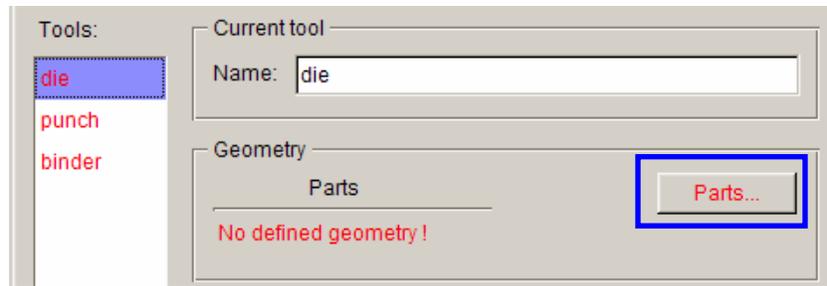


Figure 73: Tool definition page

5. The **Define Geometry** dialog box illustrated in Figure 74 is displayed. Click the **Add Part...** button.



Figure 74: Define geometry dialog box

6. Select **LOWTOOL** part in the **Select Part** dialog box illustrated in Figure 75.
7. Click **OK** to return to **Define Geometry** dialog box. The **LOWTOOL** part is now added to the list of die, as illustrated in Figure 76.
8. Click **EXIT** button to return to **Tools** definition page. You notice the tag color of **die** is changed from red to black, as illustrated in Figure 77.

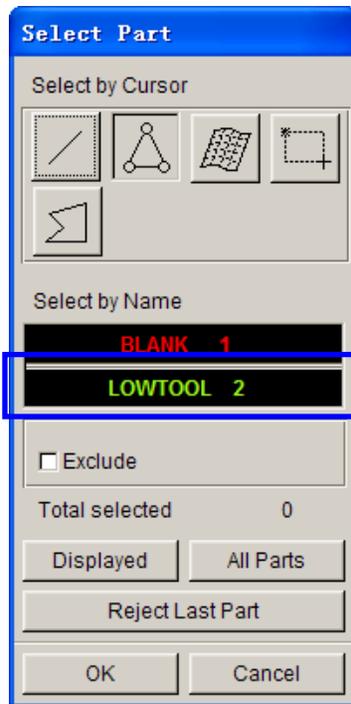


Figure 75: Select part dialog box

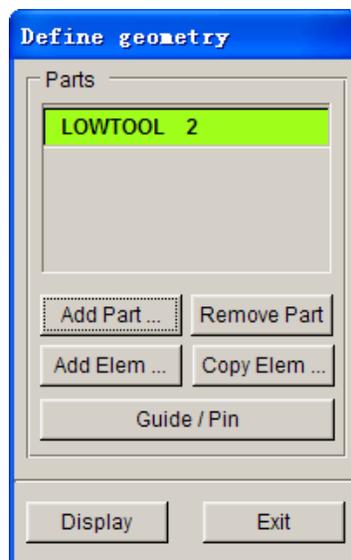


Figure 76: Define geometry dialog box

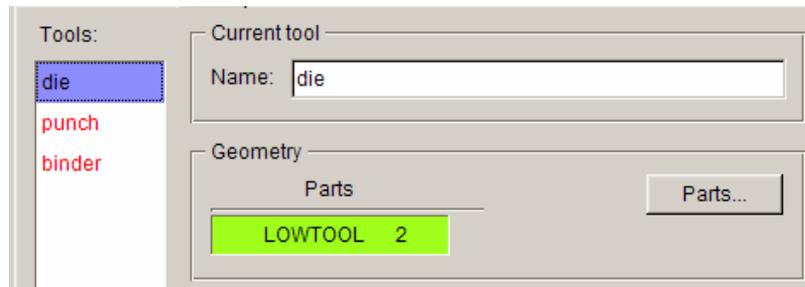


Figure 77: Tools definition page

9. Click **punch** located at the left side of tools list to display **punch** definition page. See Figure 77.
10. Click **Parts...** button in the **punch** definition page to display the **Define Geometry** dialog box.
11. Click **Copy Elem...** button in the **Define Geometry** dialog box.
12. The **Copy Elements** dialog box illustrated in Figure 79 is displayed. Then, click **Select...** button to display the **Select Elements** dialog box.



Figure 78: Define geometry dialog box

13. Select elements for **punch** from the current database. **AUTOSETUP** will copy these elements into a default part, and automatically add the part to the list of **punch**.
14. Click **DISPLAYED** button to select all elements in the **Select Elements** dialog box. Now, the program indicates 784 elements have been selected. See Figure 80.

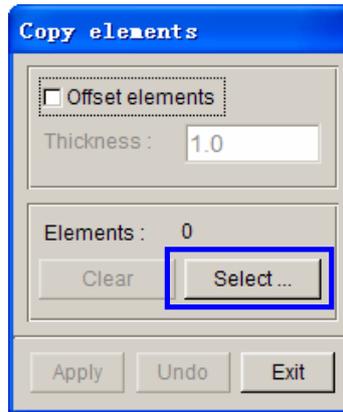


Figure 79: Copy elements dialog box

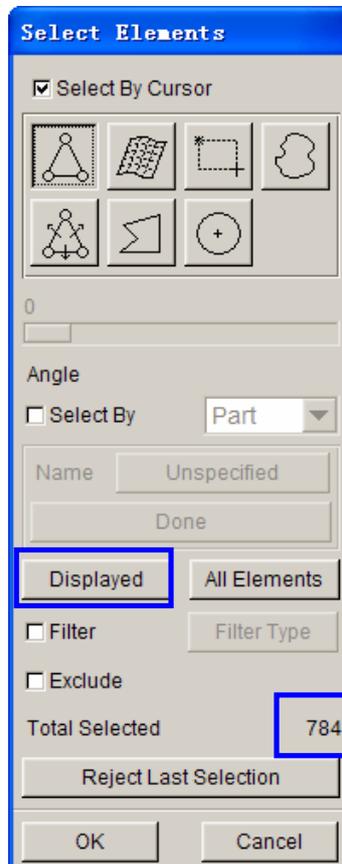


Figure 80: Select elements dialog box

- Next, you need to remove elements on the flanges of the part. Toggle on **EXCLUDE** option (shown in Figure 81). Then, click the **Surface** icon in the **Select Elements** dialog box.

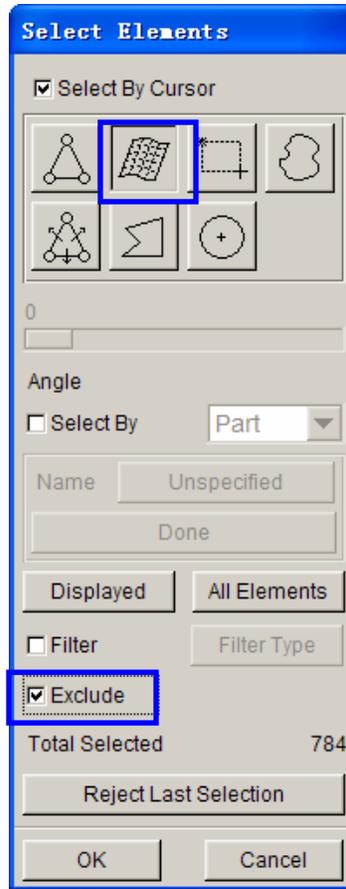


Figure 81: Select elements dialog box

16. Select two sides of the flanges. As shown in Figure 82, the highlighted elements is copied as punch.

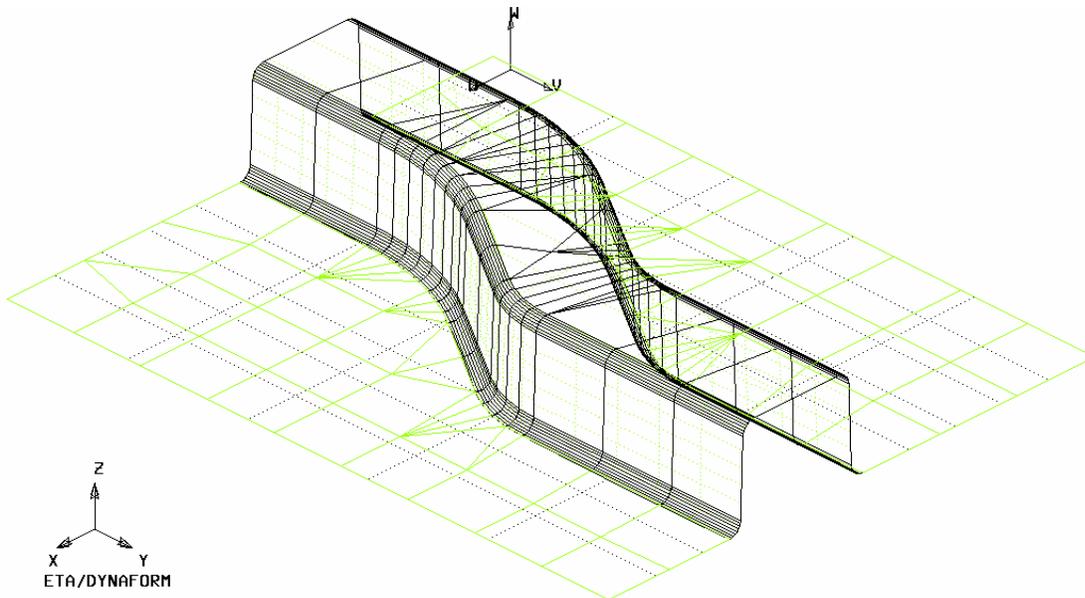


Figure 82: Highlighted elements for punch

17. The program indicates you have selected 721 elements as punch.
18. Click **OK** button to confirm the selection. The program returns to **Copy Elements** dialog box. Now, the **APPLY** button is activated. See Figure 83.

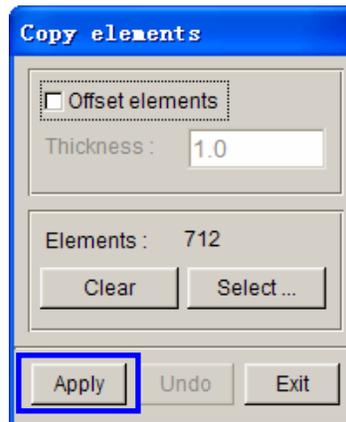


Figure 83: Copy elements dialog box

19. Click **Apply** button, to copy the elements into a new part.

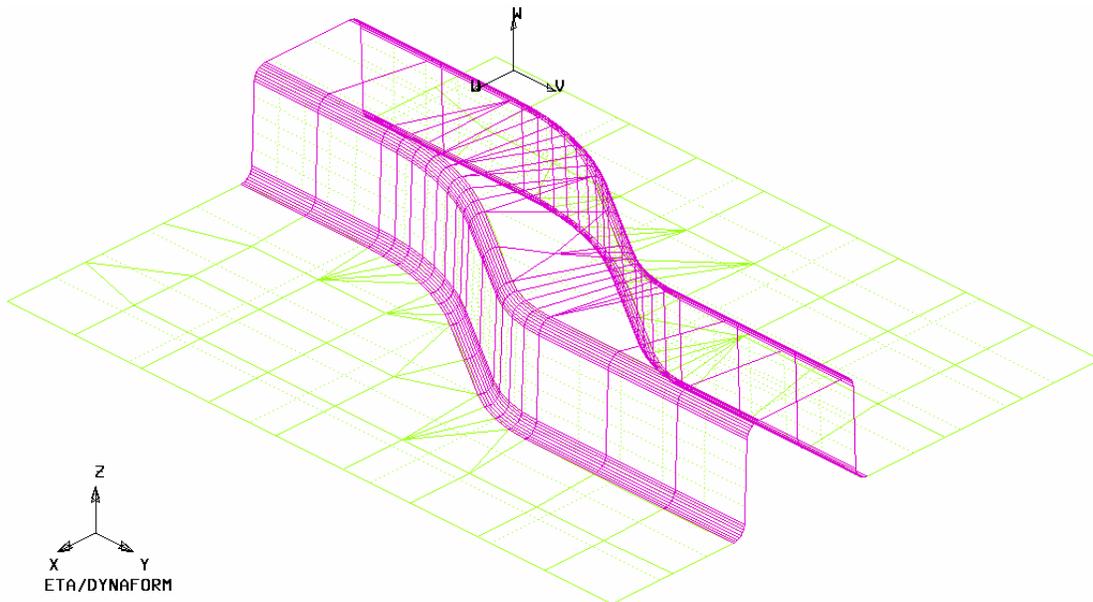


Figure 84: Create punch

20. Click **Exit** button to dismiss the **Copy Elements** dialog box. Now, a new part named **OFFSET00** is added to the list of tools geometry. See Figure 85.

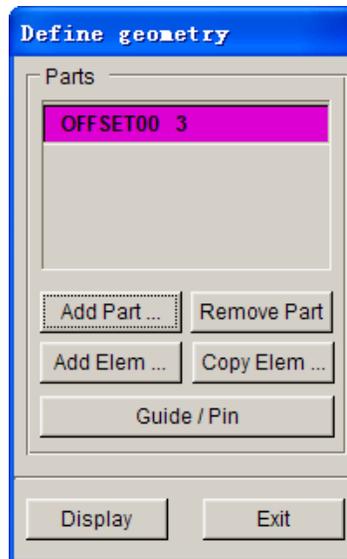


Figure 85: Define geometry dialog box

21. Click **Exit** button to return to the **Tools** definition page illustrated in Figure 86. The tag color of **punch** is changed from red into black indicating the **punch** has been defined.

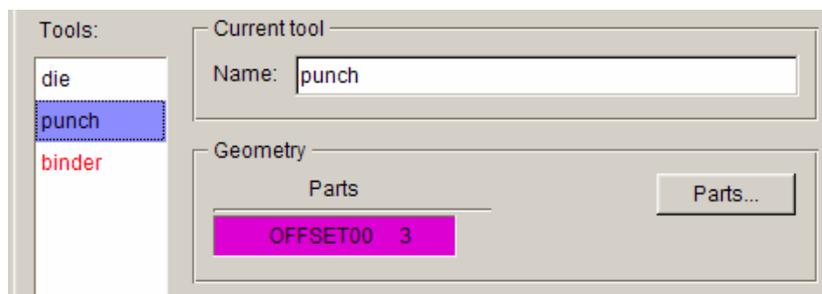


Figure 86: Punch definition page

22. Click **binder** located at the left side of tools interface to enter the **binder** definition page. Then, click **Parts...** button to display the **Define Geometry** dialog box.
23. Click **Copy Elem...** button to display the **Copy Elements** dialog box. Next, click **Select...** button to select elements on the flanges.
24. Select the flange in the two sides of tool by way of selecting surface. The program indicates 72 elements have been selected. See Figure 87.
25. Exit the **Select Elements** dialog box. Click **Apply** button to copy elements in the **Copy Elements** dialog box. All selected elements will be automatically copied to a new part, and the program will add the part to the list of **binder**. Then, click **Exit** button.
26. Click **Exit** to return the main **Tools** definition page. The tag color of all tools has been changed red into black, indicating all tools have been defined.

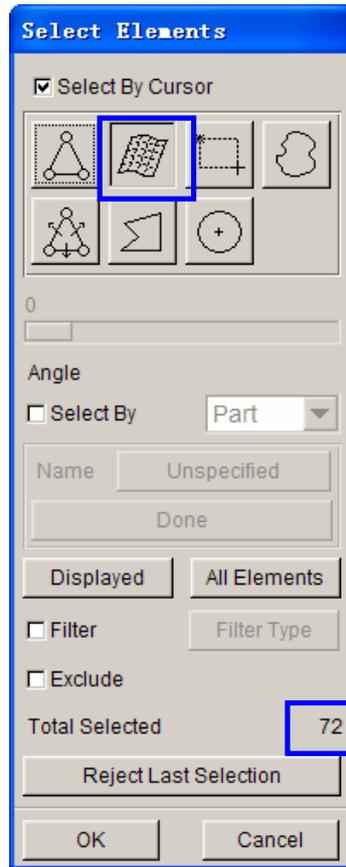


Figure 87: Select elements dialog box

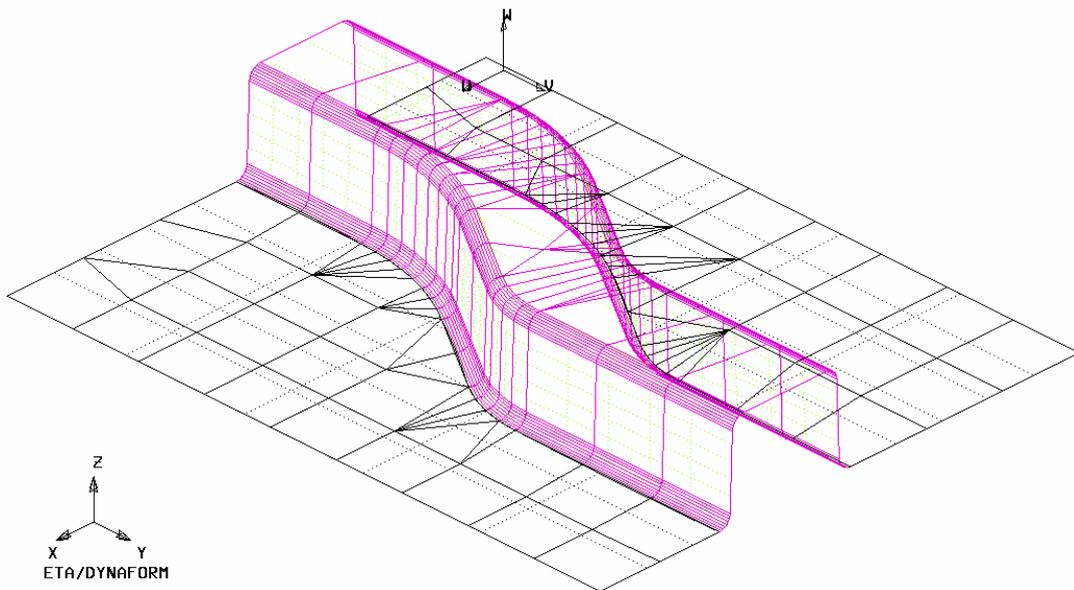


Figure 88: Highlighted elements as binder



Figure 89: Define geometry dialog box

## V. TOOL POSITIONING

After defining all tools, the user needs to position the relative position of tools for the stamping operation. The tool positioning operation must be carried out each time the user sets up a stamping simulation model. Otherwise, the user may not obtain correct stamping simulation setup.

The tool position is related to the working direction of each tool. Therefore, you need to carefully check the working direction prior to positioning the tools. If the process template is chosen, the default working direction is selected. You can also define special working direction.

1. Click **Positioning...** button located at the lower right corner of tools interface to display the **Positioning** dialog box illustrated in Figure 90. Alternatively, you may select **Tools->Positioning...** on the menu bar to display the dialog box.

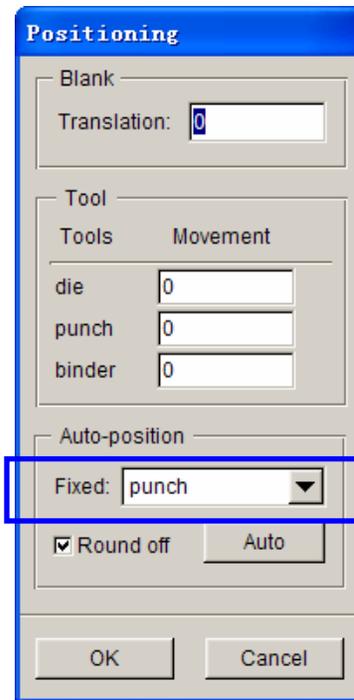


Figure 90: Positioning dialog box

2. Select **punch** as fixed tool and use it as the referenced tool in positioning operation.
3. Click **Auto** button to automatically position all tools and blanks.
4. Now, all tools and blank(s) are moved to a preset position. The value behind every tools is the distance from the home position to the current position. See Figure 91.

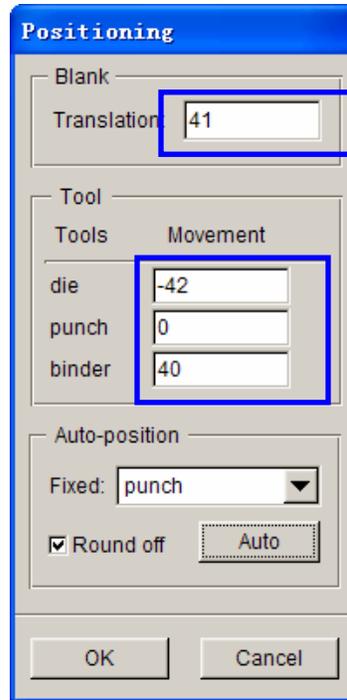


Figure 91: After positioning

5. Toggle off the **Lines** and **Surfaces** option in the **Display Options** located at the lower right corner of screen. See Figure 92.



Figure 92: Display options

6. Click  and  icon in the **Icon bar** to display the model after positioning. See Figure 93.

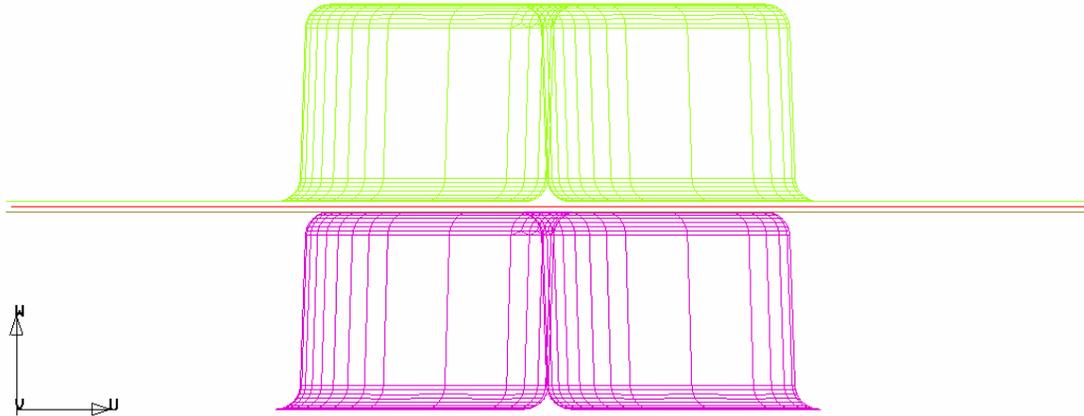


Figure 93: Model after autopositioning

7. Click **OK** button to save the current position of tools, and returns to the main interface.

## VI. PROCESS DEFINITION

The Process definition is utilized to set up process parameters of the current simulation such as time required for every process, tooling speed, binder force and so on. The parameters listed in process definition interface depend on the pre-selected process template, including type of simulation and process.

In this example, the type of simulation and process is **Sheet Forming** and **Single Action**, respectively. Therefore, two default processes are listed in the process definition interface: closing and drawing process.

You can press **Process** tab of the main interface to enter process definition interface. Next, follow the steps listed below:

1. Select **closing** process from the list located at left side of the interface as the current process. See Figure 94.
2. Verify the default setting of **closing** stage is similar to Figure 94.
3. Select **drawing** process from the list located at left side of the interface as the current process. See Figure 95.
4. Verify the default setting of **drawing** stage is similar to Figure 95.

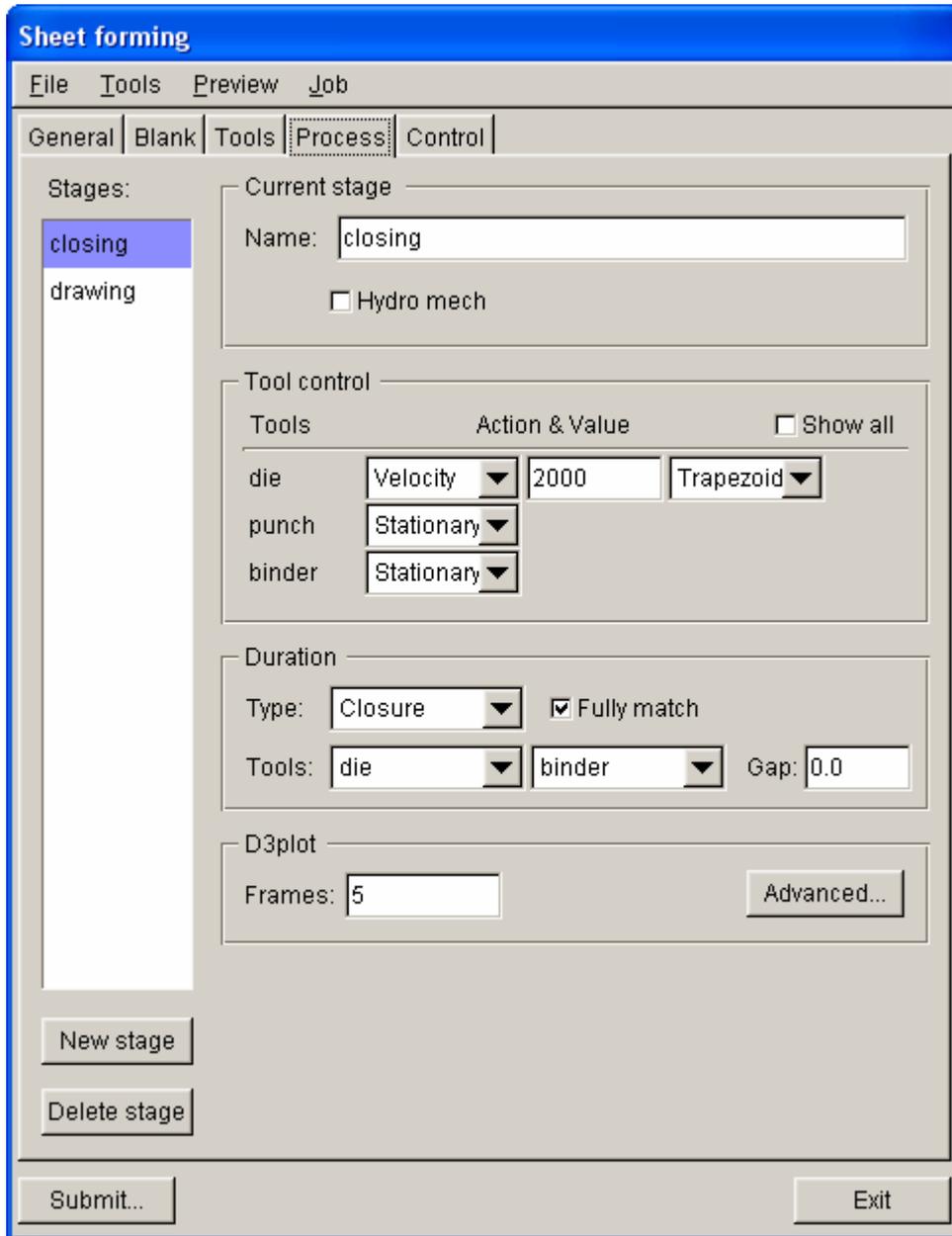


Figure 94: Closing process interface

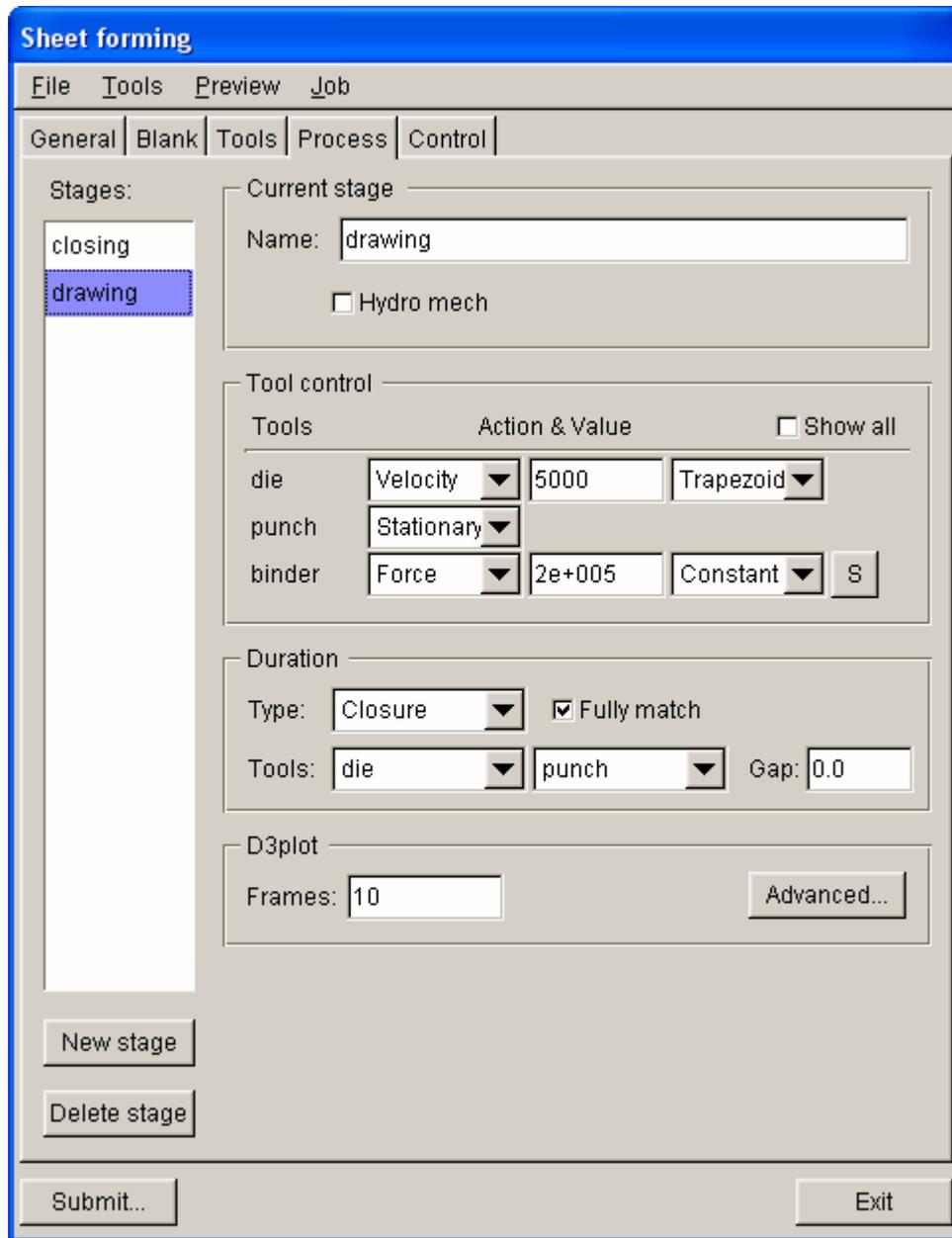


Figure 95: Drawing process interface

## VII. ANIMATION

Prior to submitting a simulation job, you shall validate the process setting by viewing the animation of tool movement. The procedure to conduct animation is listed as the following:

1. Select **Preview->Animation...** in the menu bar.

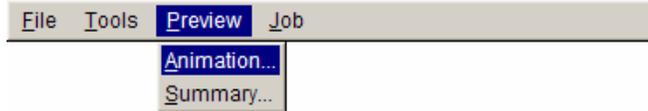


Figure 96: Animation menu

2. The animation of tool movement is shown on the display area.
3. You can select the **INDIVIDUAL FRAMES** option from the ANIMATE dialog box illustrated in Figure 97 to display the incremental tool movement.

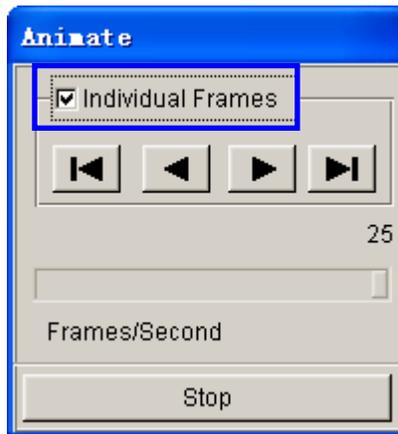


Figure 97: Animate dialog box

3. Click  icon to display the condition step by step.
4. The time step and displacement of tools are printed on the upper left hand corner of the display area. See Figure 98.
5. Click **Exit** to return the main interface.

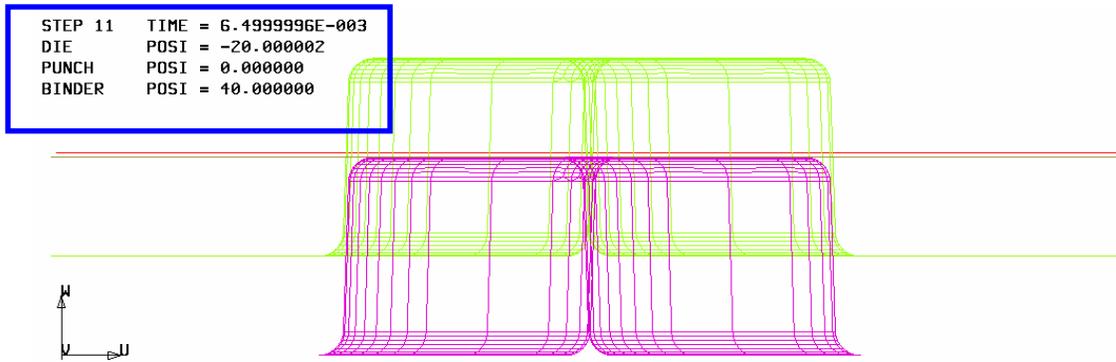


Figure 98: Animating review

## VIII. SUBMIT JOB

After validating the tools movement, you may continue with submitting the stamping simulation through the steps listed below:

1. Click **Job->Job Submitter...** in the menu bar.

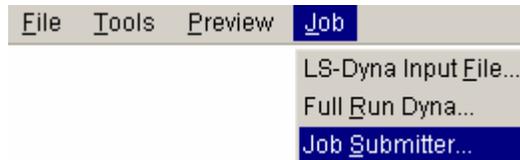


Figure 99: Job menu

2. The **Submit Job** dialog box illustrated in Figure 100 is displayed.

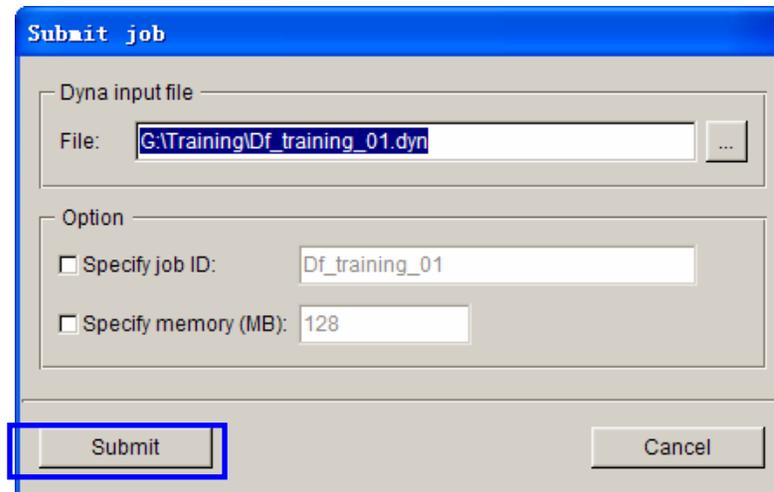


Figure 100: Submit job dialog box

3. Click **Submit** button to submit job.

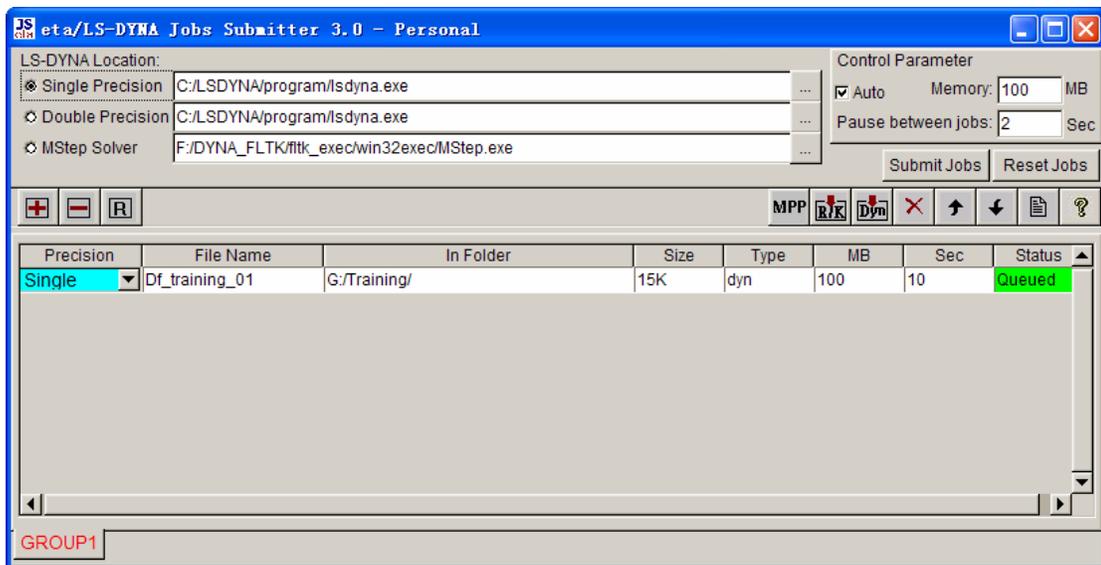
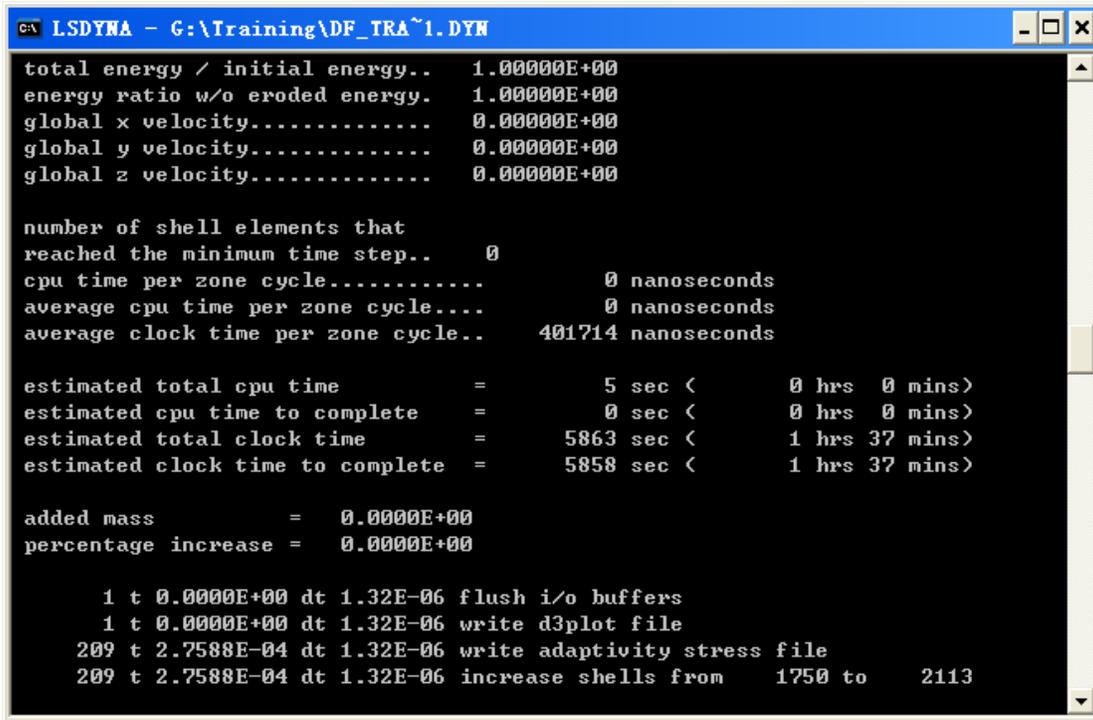


Figure 101: Job Submitter interface

4. Click **Submit Jobs** button, as shown in Figure 101 to activate the LS-DYNA solver for running the stamping simulation. The LS-DYNA window illustrated in Figure 102 is displayed and the simulation is in progress.



```
c:\ LSDYNA - G:\Training\DF_TRA~1.DYN
total energy / initial energy.. 1.00000E+00
energy ratio w/o eroded energy. 1.00000E+00
global x velocity..... 0.00000E+00
global y velocity..... 0.00000E+00
global z velocity..... 0.00000E+00

number of shell elements that
reached the minimum time step.. 0
cpu time per zone cycle..... 0 nanoseconds
average cpu time per zone cycle.... 0 nanoseconds
average clock time per zone cycle.. 401714 nanoseconds

estimated total cpu time      =      5 sec <      0 hrs  0 mins>
estimated cpu time to complete =      0 sec <      0 hrs  0 mins>
estimated total clock time    =    5863 sec <      1 hrs 37 mins>
estimated clock time to complete =    5858 sec <      1 hrs 37 mins>

added mass      =  0.0000E+00
percentage increase =  0.0000E+00

  1 t 0.0000E+00 dt 1.32E-06 flush i/o buffers
  1 t 0.0000E+00 dt 1.32E-06 write d3plot file
209 t 2.7588E-04 dt 1.32E-06 write adaptivity stress file
209 t 2.7588E-04 dt 1.32E-06 increase shells from 1750 to 2113
```

Figure 102: LS-DYNA solver window

# TRADITIONAL SETUP

Open the saved database as described in meshing section of this manual (on page 22). This file should contain the clean mesh data of the lower tool before it was separated for the QuickSetup procedure. If you do not have this database, repeat the procedure of **MESHING**.

## I. OFFSETTING THE LOWER TOOL MESH TO CREATE THE UPPER TOOL

1. Create a new part called **UPTOOL**. This part will keep the elements offset from the **LOWTOOL**. Click **Parts**→**Create on Menu bar**.
2. In the **New Part** dialog box, enter “**UPTOOL**” in the name field. Click **OK** button to create the part.

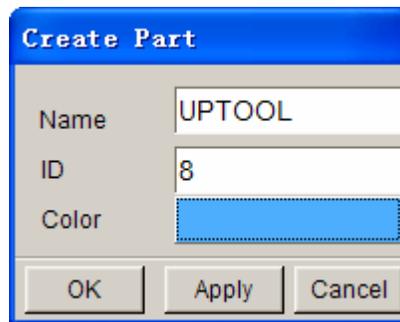


Figure 103: Create part dialog box

The part **UPTOOL** has been created and set as the current part automatically. You can now offset elements into this part. Turn on **LOWTOOL** and turn off **BLANK.LI**.

3. Select **Preprocess**→**Elements** on the **Menu bar**.

<u>L</u> ine/Point	Ctrl+L
<u>S</u> urface	Ctrl+S
<u>E</u> lement	Ctrl+E
<u>N</u> ode	Ctrl+N
<u>M</u> odel Check/Repair	Ctrl+R
<u>B</u> oundary Condition	Ctrl+U
<u>N</u> ode/Element <u>S</u> et	Ctrl+V

Figure 104: Preprocess menu

- Select the **Copy Elements** icon from the **Element** dialog box illustrated in Figure 105.



Figure 105: Element dialog box

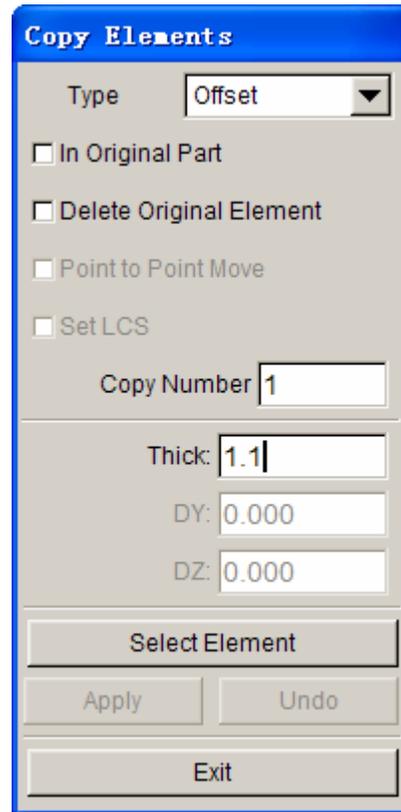


Figure 106: Copy elements dialog box

- Click the drop-down menu to the right side **Type**, and select **Offset**. See Figure 106.
- Toggle off **In Original Part** so that the offset elements will be put in the current part. Make sure **UPTOOL** is set as current. Toggle off **Delete Original Part**, the original part will be kept.
- Enter **1** in the **Copy Number** field as the number of copies. Enter **1.1** (mm) in the **Thick** field as the offset thickness. In eta/DYNAFORM, the offset thickness is based on the material thickness of the blank plus 10% tolerance.
- Click **Select Element** button.
- Select Elements** dialog is displayed. Click the **Displayed** button illustrated in Figure 107. You notice that the selected elements are highlighted in white.

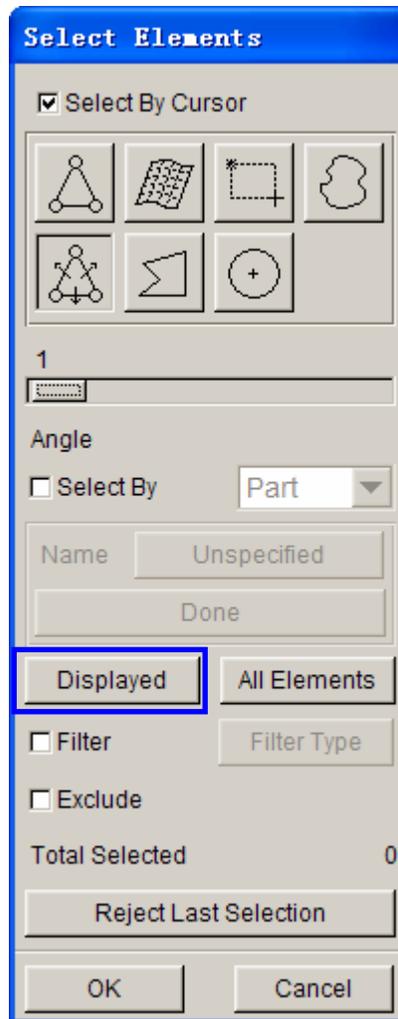
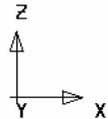
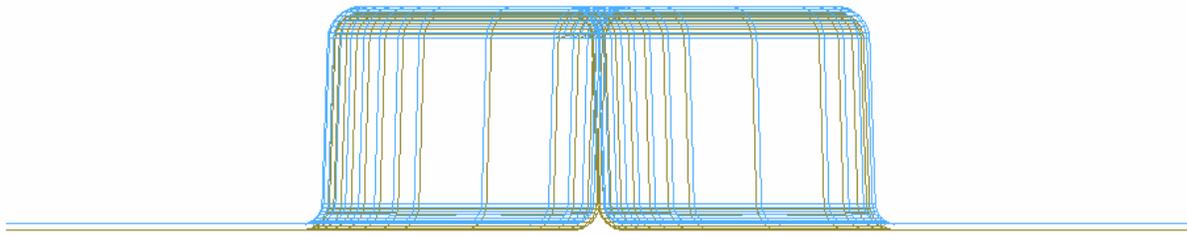


Figure 107: Select elements dialog box

10. Click **OK** to accept the selected elements and return to the **Copy Elements** dialog box.
11. Click on **Apply** button. All offset elements are placed into the part **UPTOOL**. Check your display by clicking the **Left View** icon. See Figure 108.



ETA/DYNAFORM

Figure 108: Model

12. If your results differ, select **Undo**. Repeat the above steps to create mesh of **UPTOOL**.
13. Turn off the part **LOWTOOL** so that only **UPTOOL** is displayed. Switch to the isometric view.
14. Save your database.

## II. CREATE THE LOWER RING

Now, you will separate the **Lower Ring** from the **LOWTOOL** and move elements on Run-out of **LOWTOOL** into **Lower Ring**.

1. Turn on **LOWTOOL** and turn off all other parts.
2. Create a new part called **LOWRING**. This part will hold the elements that we separate from the **LOWTOOL**. Click **Parts** → **Create** on **Menu bar**.
3. Enter **LOWRING** in the name field. Click **OK** and the part will be created.

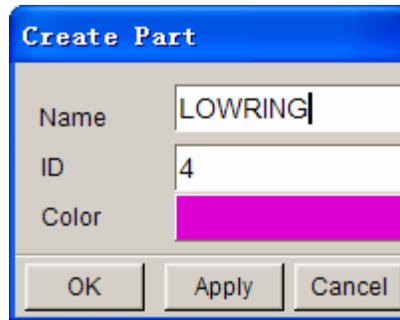


Figure 109: Create part dialog box

4. The part **LOWRING** has been created and set as the current part automatically. You place the lower ring elements in this part.
5. Click on **Parts** ➔ **Add...To Part** from **Menu bar**.
6. The **Add...To Part** window appears as follows. Click the **Element(s)** button as shown below.

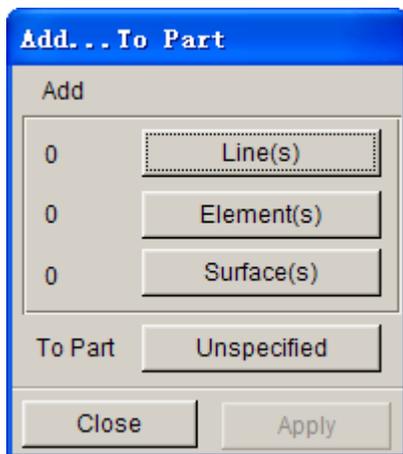


Figure 110: Add...To Part dialog box

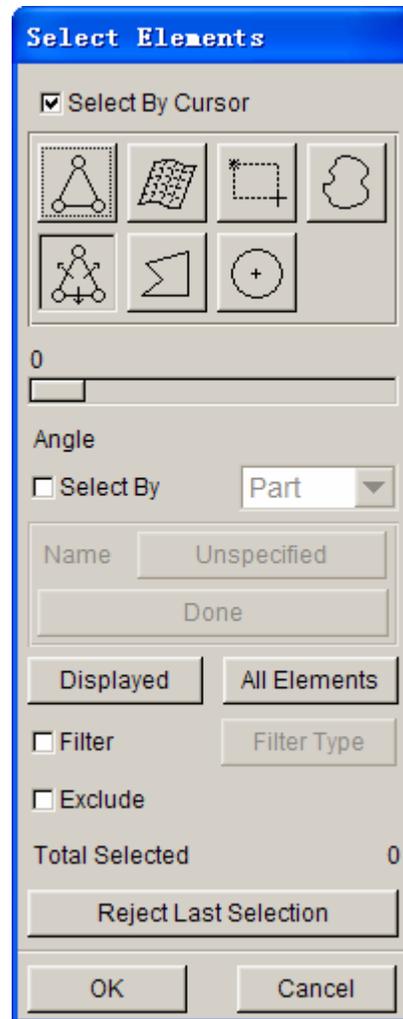


Figure 111: Select elements dialog box

7. The **Select Elements** window appears (as shown above). The easiest way to select all elements of the ring is to switch the view to the **Top View (YX plane)**. Then, select the **Spread** icon. Click the slider and drag it to set an angle. Since the ring surface is flat, set a small angle, such as 1 degree.
8. Click any element on the right ring of **LOWTOOL**.
9. Click any element on the left-ring of **LOWTOOL**.

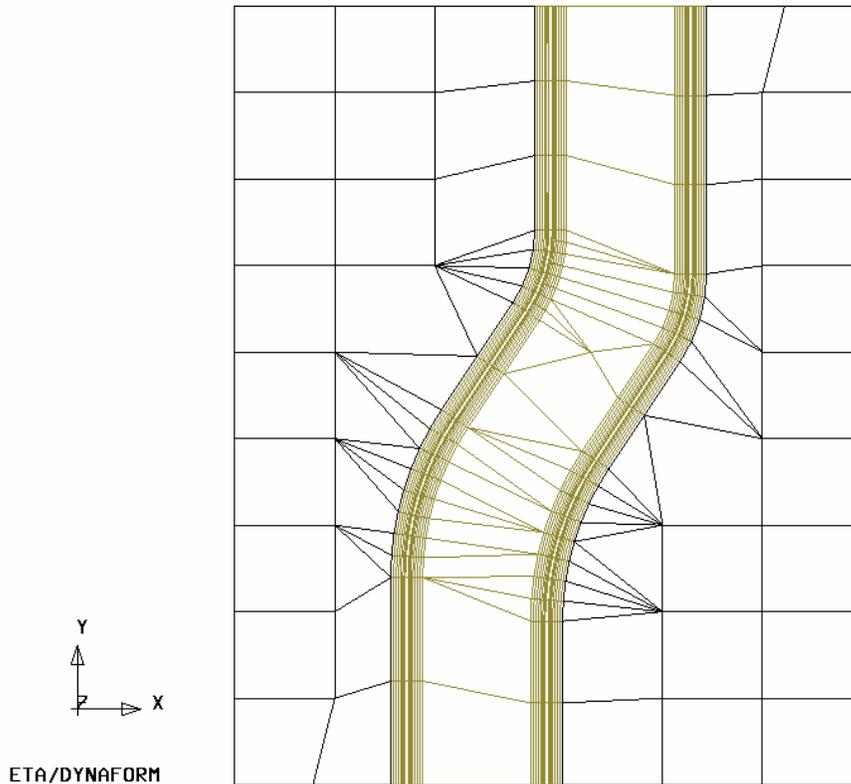


Figure 112: Highlighted elements

All elements in the flat area before an element angle change of larger than 1 degree should be highlighted. Compare your display to the preceding image. If your results differ, repeat the above steps.

10. Select **OK** on the **Select Elements** window. You will find the number of elements (67) is shown on the left side of the **Element(s)** button as below.
11. Select the **Unspecified** button.

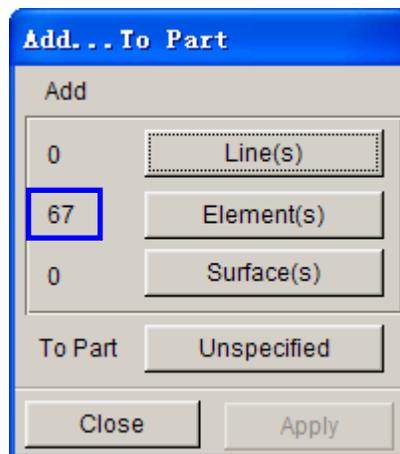


Figure 113: Add...To Part dialog box

12. The **Select Part** window shows up. Select **LOWRING** in the **Select by Name** list. At this time, the **Unspecified** button will change to **LOWRING**.
13. Click **Apply**, all selected elements are moved into **LOWRING**.
14. Turn on only **LOWRING** and display the part using **Isometric view**, the resulting display should be as the following. If the result differs, please repeat the above steps.

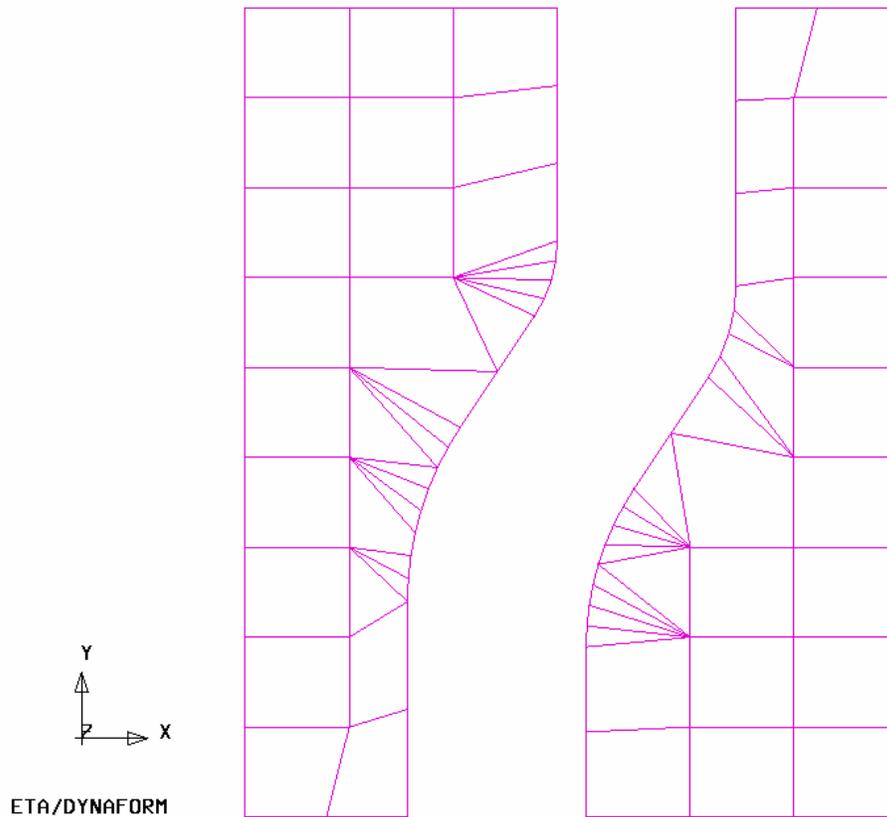


Figure 114: Illustration of binder

15. Save the database.

### III.SEPARATE LOWRING AND LOWTOOL

Now **LOWRING** and **LOWTOOL** have two different groups of elements but they share common nodes along the boundaries. We need to separate them so that **LOWRING** and **LOWTOOL** can move independently.

Let's turn on **LOWRING** and **LOWTOOL** and turn off other parts.

1. Click **Parts**⇒**Separate** on the **Menu bar**.

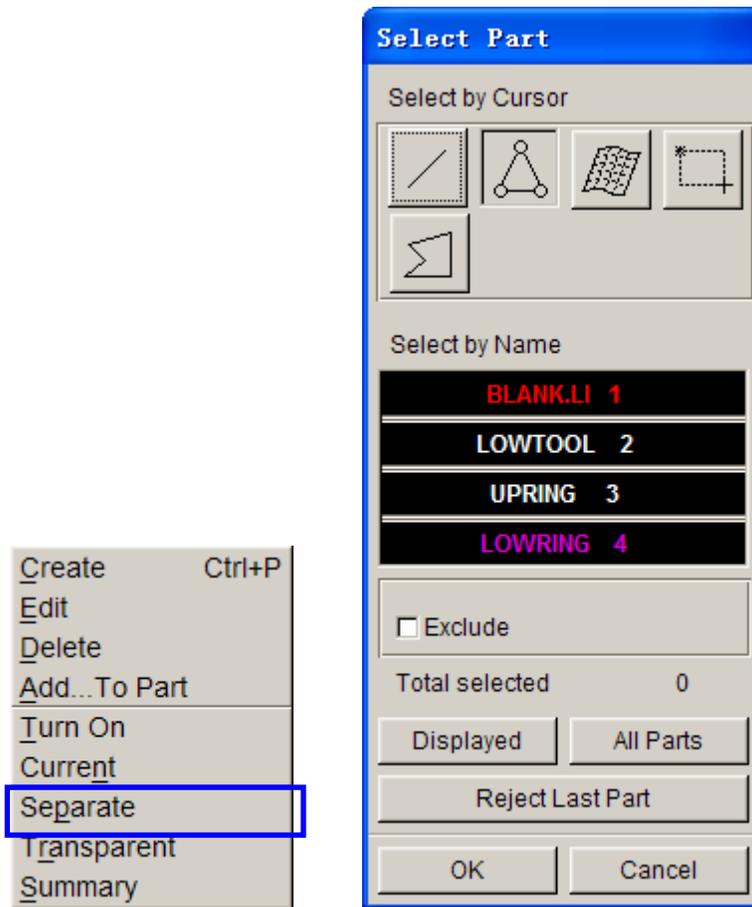


Figure 115: Separate function

2. The **Select Part** window appears. Select **LOWTOOL** and **LOWRING** in the **Select by Name** list. Click **OK** to finish the function.
3. Save the changes.

#### IV. DRAW TYPE SETUP

Prior to defining the tools, we first setup the draw type.

Click **Tools**→**Analysis Setup** from Menu bar. Select the Single Action as the **Draw Type**.

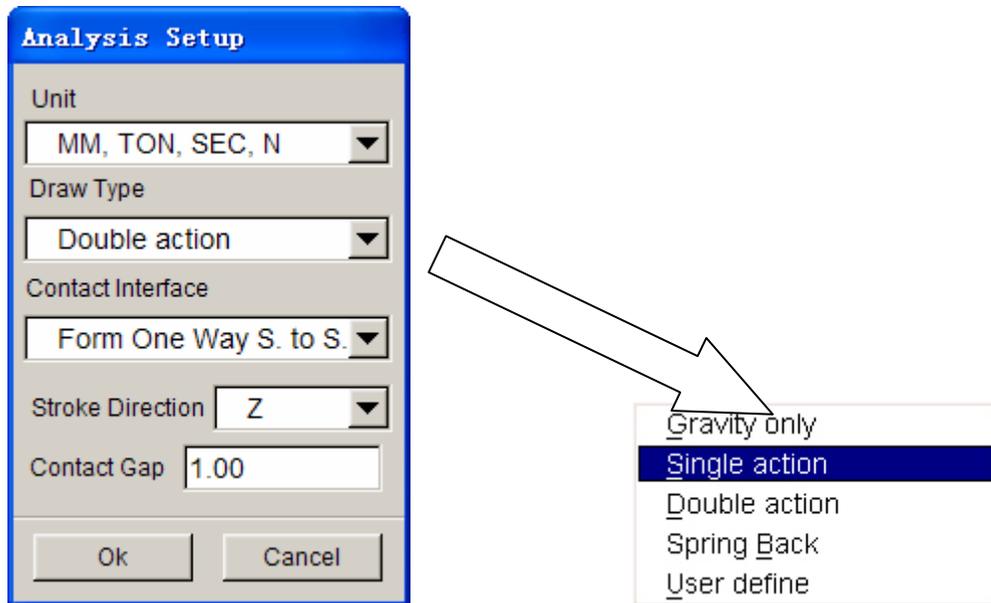


Figure 116: Analysis setup dialog box

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**. You can also refer to the **eta/DYNAFORM User's Manual** for information on the **draw type**.

## V. TOOL DEFINITION

### Defining Parts as Tools

The parts **BLANK.LI**, **LOWTOOL**, **UPTOOL**, and **LOWRING** are all meshed and can now be defined as tools.

1. Select **Tools**→**Define Tools** on **Menu bar**.

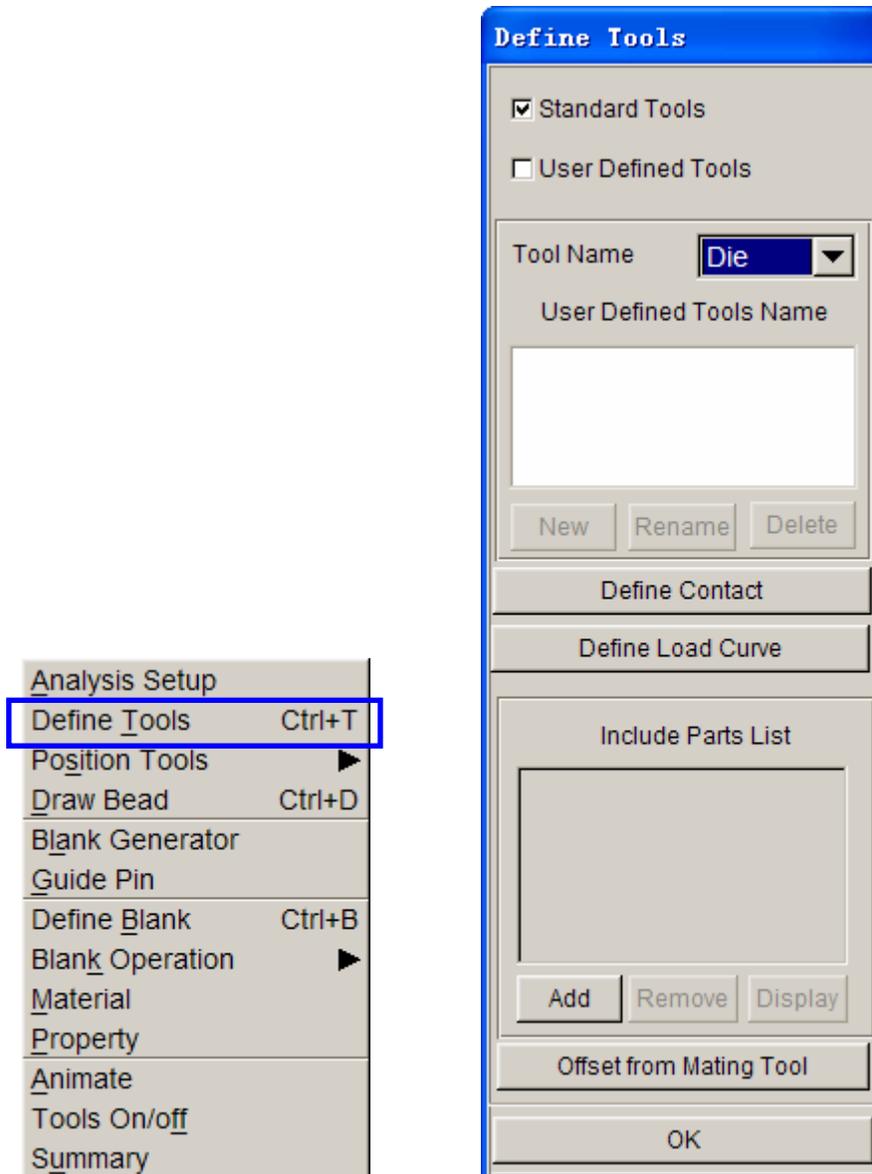


Figure 117: Define tools dialog box

2. In the **Define Tools** dialogue, select **DIE** from the **Tool Name** menu as below.
3. Select **ADD**.
4. The **Select Part** dialogue will be displayed, prompting you to select which part will be defined as the DIE. Choose **UPTOOL** from the **Select by Name list** and then click **OK**.
5. Return to the **Define Tools** dialogue. You will find the **UPTOOL** is placed in the **Include Parts List**.

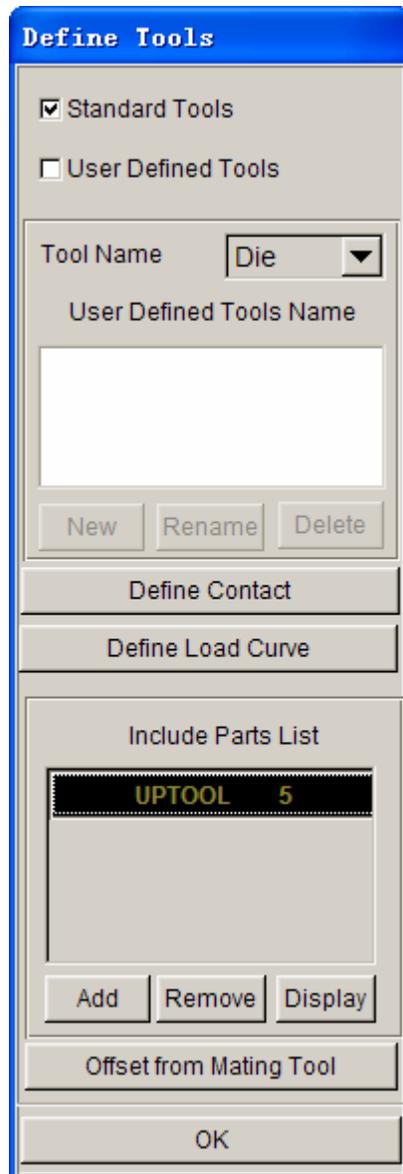


Figure 118: Define tools dialog box

- Repeat the above steps to define the Punch (**LOWTOOL**) and Binder (**LOWRING**). Remember to select the correct Tool Name in Step 2.
- Once you have all of the tools defined, click **OK** on the bottom of **Define Tools** dialogue to finish this step. Save the changes.

## VI. DEFINING THE BLANK AND SETTING UP PROCESSING PARAMETERS

### Define the Blank

1. Select **Tools**⇒**Define Blank** on the Menu bar.

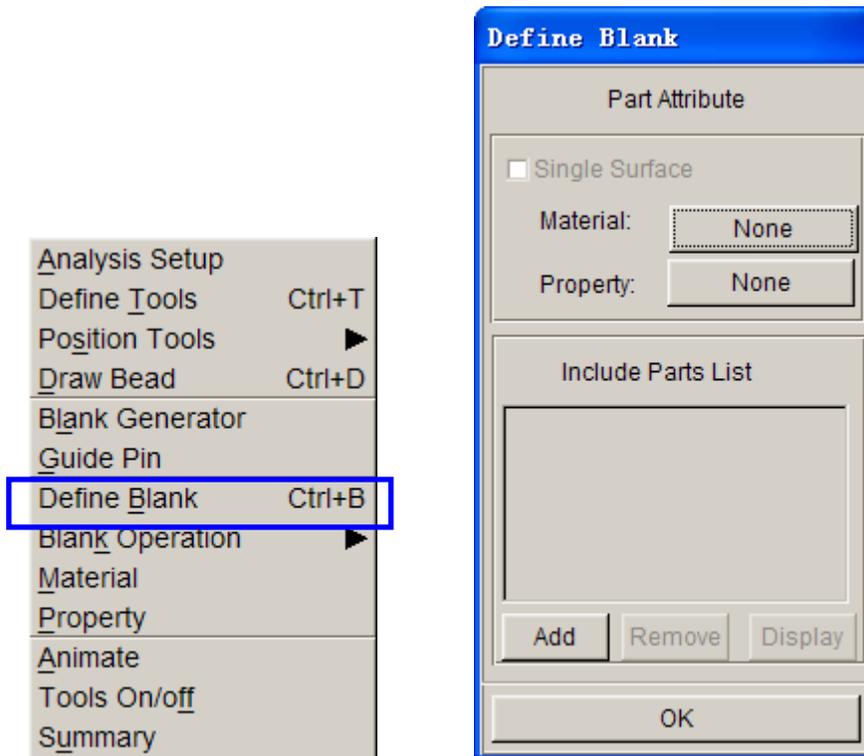


Figure 119: Define blank dialog box

2. Click the **Add** button in the **Define Blank** dialog box.
3. The **Select Parts** dialogue is displayed. Select **BLANK.LI** from the **Select by Name** list.
4. Click **OK**. The **BLANK.LI** is added to the **Include Part List** as shown below.

### Define the Blank Material

1. The **Define Blank** dialogue should still be opened, click the button located right next to “**material:**”

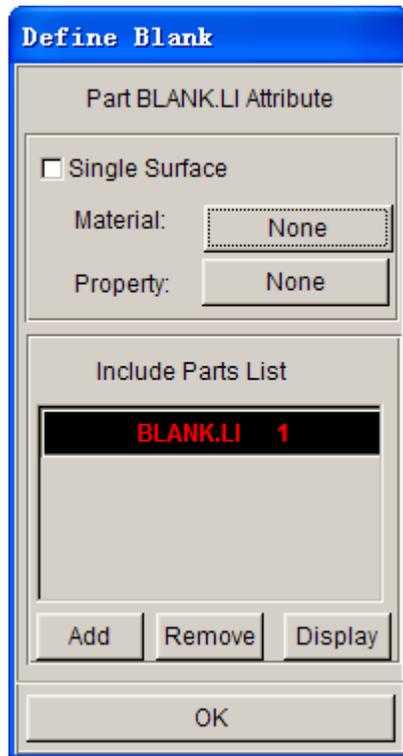


Figure 120: Define blank dialog box

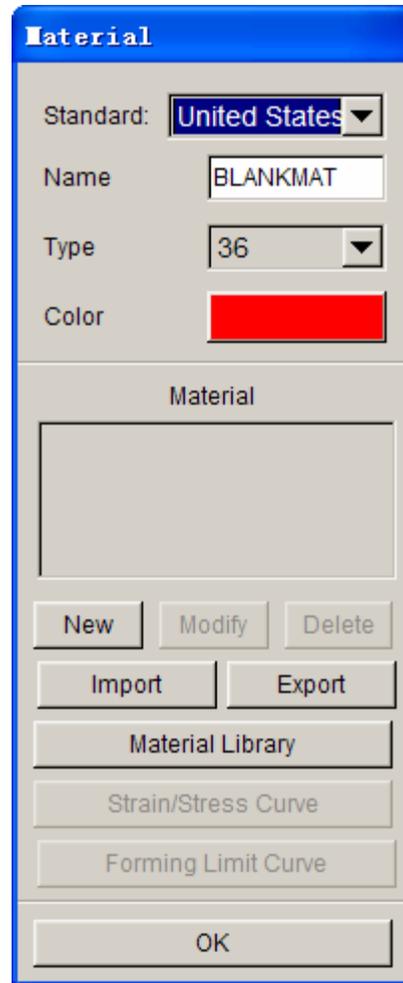


Figure 121: Material dialog box

2. Select **Material Library**. A list of materials will be shown as in the illustration below. In this dialogue, select the **mild steel “DQSK”** under material type 36 (**Note:** material type 36 and 37 are recommended for most simulations).

Dynaform Material Library									
	Strength Level	Material Name	Type 1 ELASTIC	Type 18 POWER	Type 24 LINEAR	Type 36 3-PARAM	Type 37 ANISOTR	Type 39 FLD_TRA	Type 64 RATE_SEN
STEEL	Mild	DQ	+	+	+	+	+	-	-
		DQSK	+	+	+	+	+	-	-
		DDQIF	+	+	+	+	+	-	-
		CQ	-	+	+	+	+	-	-
	Medium	BH180	+	+	+	+	+	+	-
		BH210	+	+	+	+	+	+	-
		BH250	+	+	-	+	+	+	-
		BH280	+	+	+	+	+	+	-
	High	HSLA250	+	+	+	+	+	+	-
		HSLA300	+	+	+	+	+	-	-
		HSLA350	+	+	+	+	+	+	-
		HSLA420	+	+	+	+	+	-	-
	Advanced High	DP500	+	+	+	+	+	-	-
		DP600	+	+	+	+	+	-	-
	Hot Rolled	CQ	+	+	+	+	+	-	-
		DQSK	+	+	+	+	+	-	-
		DDQIF	+	+	+	+	+	-	-
		HSLA400	+	+	+	+	+	-	-
	Stainless	SS11CrCb	+	+	+	+	+	-	-
		SS18CrCb	+	+	+	+	+	-	-
SS304		+	+	+	+	+	-	-	
SS409Ni		+	+	+	+	+	-	-	
ALUMINUM	AA5182	+	+	+	+	+	-	-	
	AA5454	+	+	+	+	+	-	-	
	AA5754	+	+	+	+	+	-	-	
	AA6009	+	+	+	+	+	-	-	

Figure 122: Generic material library window

3. Select **OK** to use the default material parameters (following image) for the DQSK material model (**Note:** ETA makes no guarantees as to the validity and/or accuracy of the generic material models from the material library. Users should contact their material suppliers to determine proper material parameters).

### Define Blank Property

1. The **Define Blank** dialog box remains, click the button just right of where it says **Property:** (This button will say **None** because the property has not been defined yet).

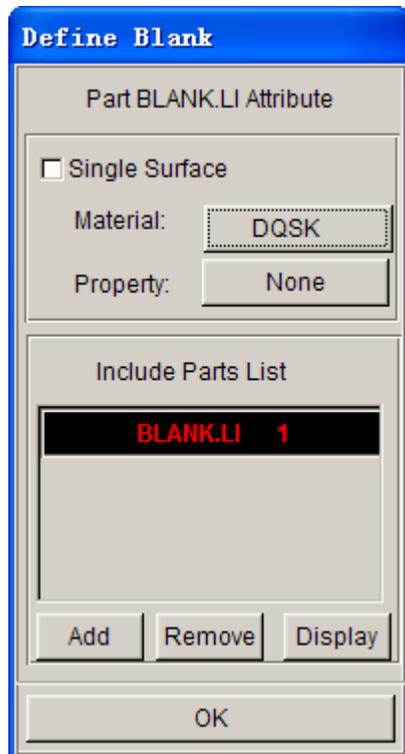


Figure 123: Define blank dialog box

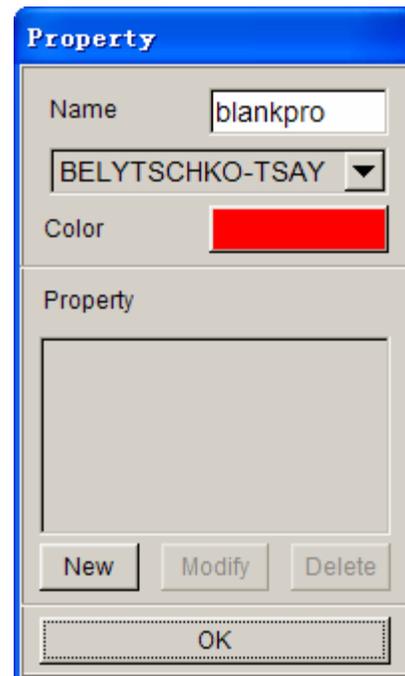


Figure 124: Property dialog box

2. In the **Property** dialogue, enter a name for the material, For example put “blankpro” for material property. (If there is a default property name exists, user can skip the typing). Be sure to select type **BELYTSCHKO-TSAY** (default) and select any color from **Color** dialogue. Click **New**.
3. The **BELYTSCHKO-TSAY Property** card will be displayed. In this dialogue, you can edit the thickness of the material. To edit a field, left click on the field and change the value in the **Current Value** dialogue. Make sure **UNIFORM THICKNESS** is 1.000. Leave all other fields at their default values and select **OK**.

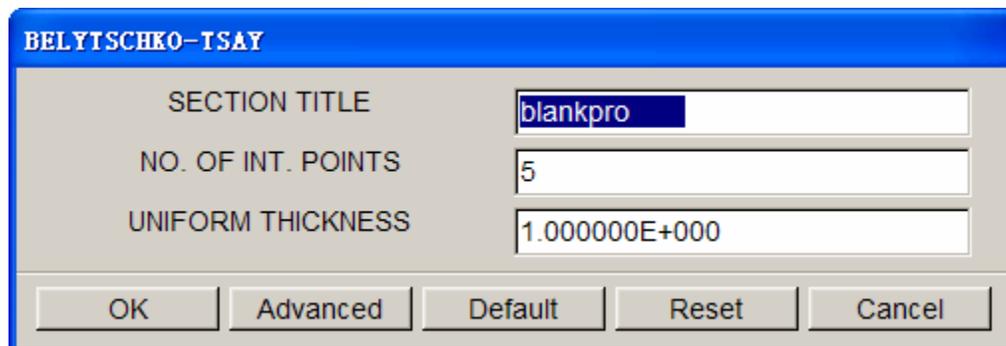


Figure 125: Belytschko-tsay dialog box

4. eta/DYNAFORM returns to the **Property** dialogue. Click **OK**.
5. Select **OK** to finish defining the Blank, Material and Property.

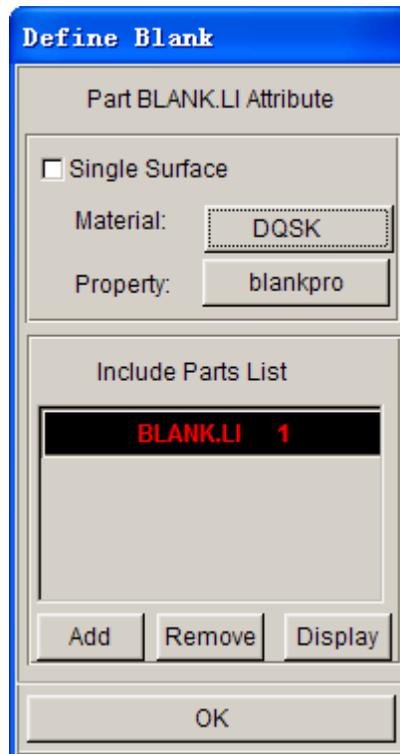


Figure 126: Define blank dialog box

## VII. TOOLS SUMMARY

1. Verify that all of the needed tools are defined by selecting the **Tools Summary** function in the **Tools** menu.
2. After you have verified the tooling definition, click **OK**. Save the changes.

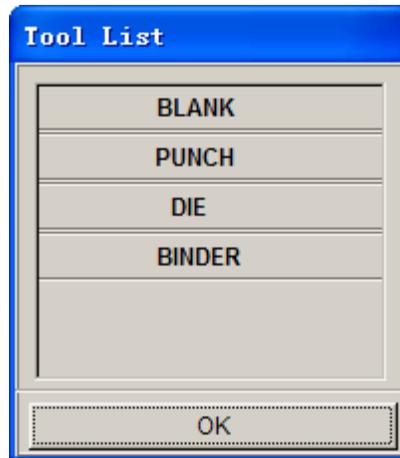


Figure 127: Tool list dialog box

## VIII. AUTO POSITIONING THE TOOLS

Now that all of the tools have been defined, we need to place them in the correct position by doing the following:

1. Turn on all of the parts in the database and select the **isometric view**.
2. Click **Tools**→**Position Tools**→**Auto Position** on the **Menu bar**.

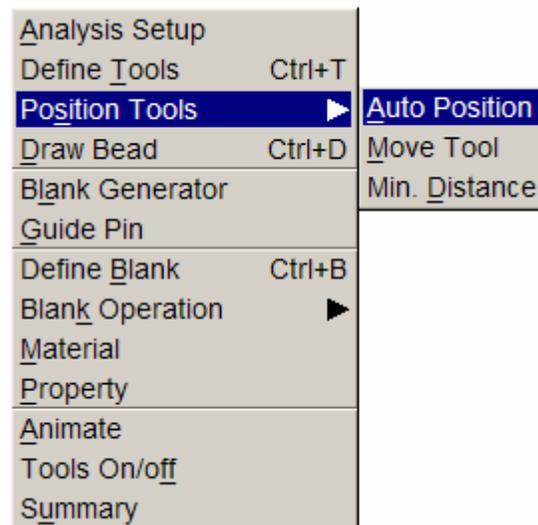


Figure 128: Position tools menu

3. The **Auto Position Tools** dialogue will be displayed. In this dialogue, define the Master and Slave Tools. The Master Tool is the tool that will not move while you Auto Position; it should be the Blank. Select **BLANK** in the **Select Master Tool** dialogue and then select the

remaining tools in the **Select Slave Tools** dialogue. (For the PC version, the user needs to hold down the control key to select all three slave tools).

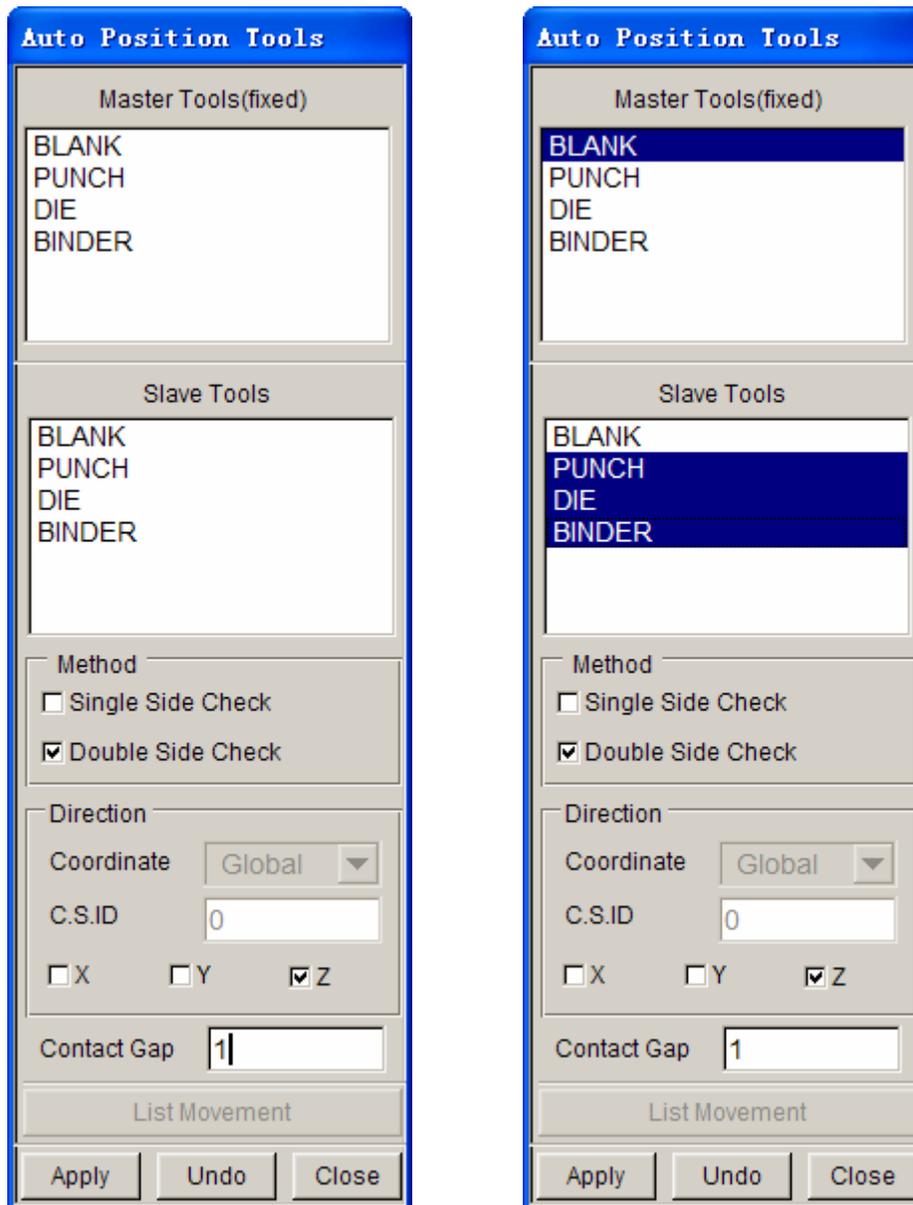


Figure 129: Autoposition tools dialog box

Once the correct Master and Slave Tools are selected, be sure that **Z** (direction) is selected as the moving direction and enter a **Contact Gap** of **1**. The gap value should be the Blank Thickness to avoid initial penetration. Press **Apply** and the tools will be automatically positioned.

4. Compare your display with the following image.

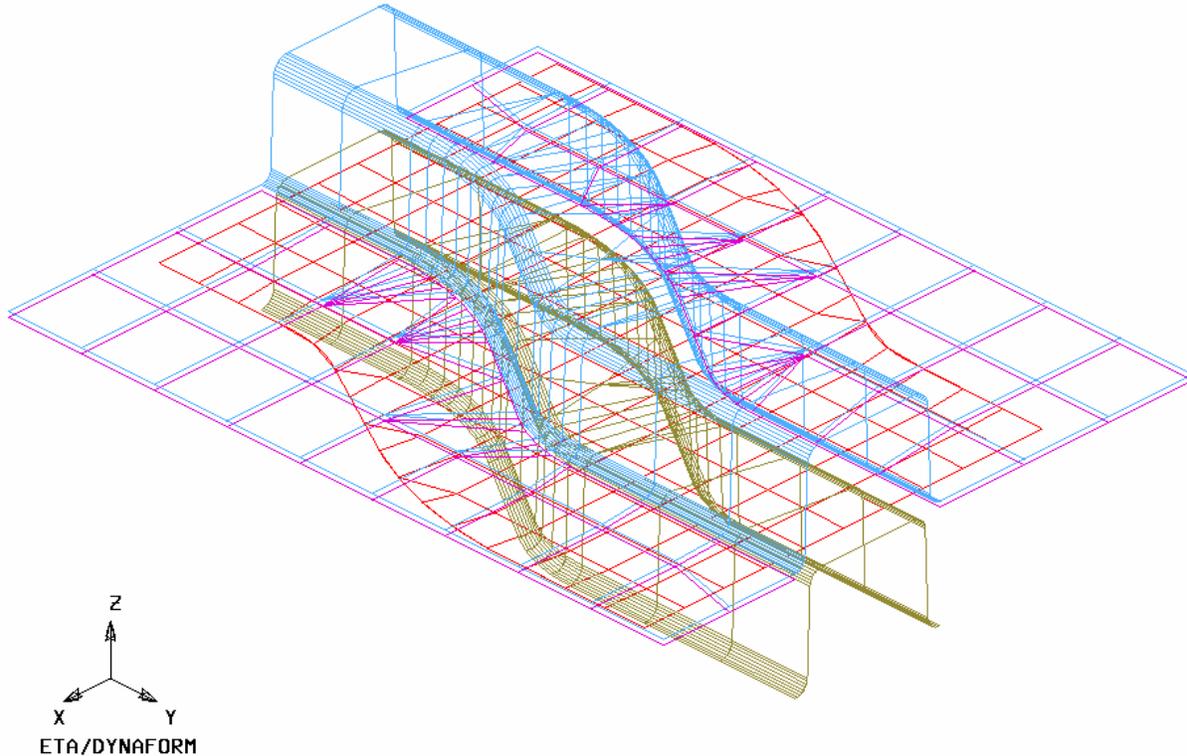


Figure 130: Illustration of model

5. If your result differs, select **UNDO** to repeat the above steps and make sure the result is correct. If the position is wrong, check **Tools/Analysis Setup**, make sure the “Draw type” is set to inverted.
6. Select **CLOSE** to exit the auto position menu.
7. Save the changes.

## IX. MEASURING THE DIE TRAVEL DISTANCE

Now that the parts have been meshed and defined as tools, we can setup the motion curves. The first step is to find the travel distance of the tools.

1. Click **Tools**⇒**Position Tools**⇒**Min Distance** on **Menu bar**.

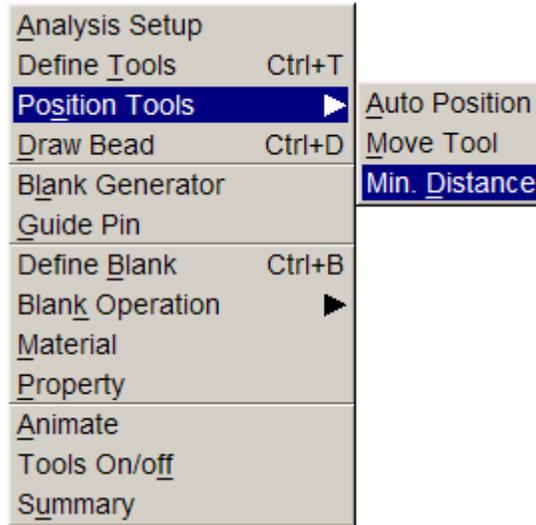


Figure 131: Min. Distance function

2. The **Min. Distance** dialogue will be displayed. Select **Z** (direction) as the direction to measure in, then highlight the **DIE**, following by the **PUNCH**. This will allow user to measure the distance, in the Z-direction, between the Punch and the Die, and display it in the **Distance** dialogue.

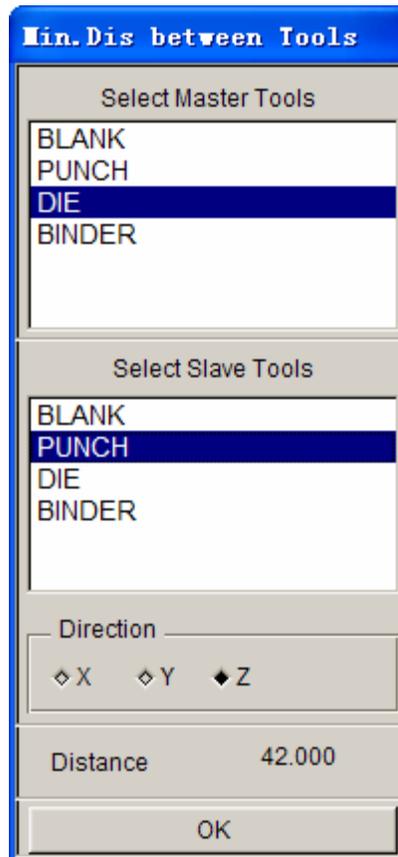


Figure 132: Min. Dis between tools dialog box

3. The total distance between the Punch and the Die is approximately 42. To find the Punch travel distance, subtract the offset thickness (blank thickness + 10%). We will obtain a Punch travel distance of approximately 40.9. Take note of the number you calculated.
4. Select **OK** located at the bottom of the **Min. Distance** dialogue, and end this step.

The automatic distance measurement is reliable for flat binders. For more complicated binder shapes, the user should measure the travel distance using the distance measurement tool between nodes found under the Utilities menu.

## **X. DEFINE DIE VELOCITY CURVE**

1. Click **Tools**→**Define Tools** on the **Menu bar**.
2. Select Die from the **Tool Name** list and then select the **Define Load Curve** button.

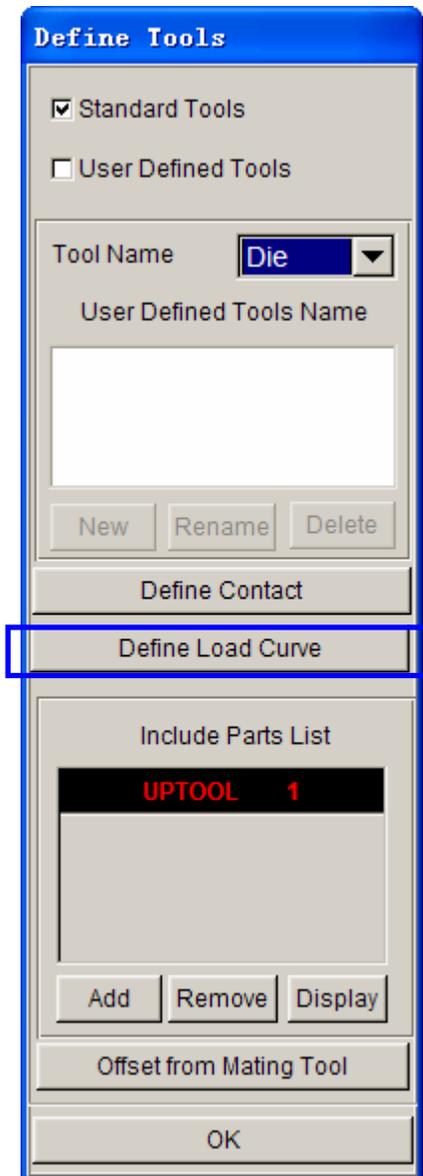


Figure 133: Define tools dialog box

Toggle the Z button on to free the Z direction travel

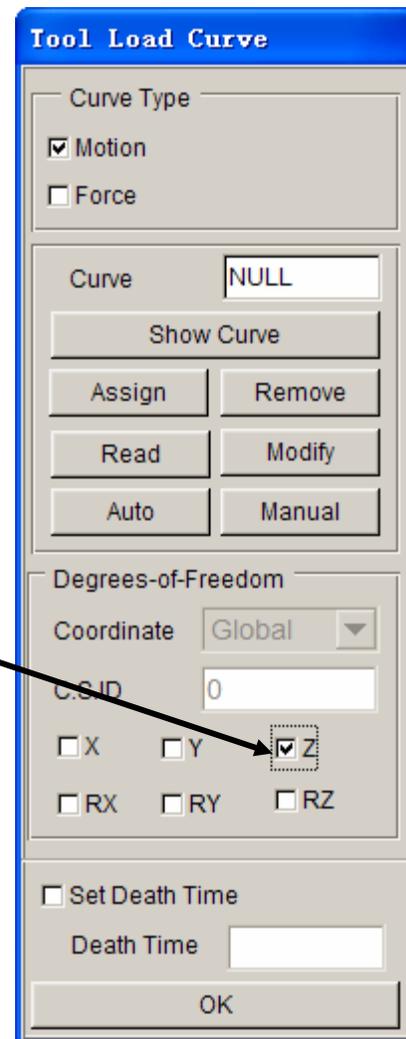


Figure 134: Tool load curve dialog box

3. Click on **Auto**. Make sure the **Z** is selected as the Degrees-of-Freedom to define the moving direction. Use the default curve type (**Motion**) (see the following illustration).
4. Choose Velocity as the default **Curve Type** (see illustration below) and default **Curve Shape** as **Trapezoidal**. Keep the beginning time of 0.000E+000. In the **Velocity** field, enter 8000 (mm/s). For the **Stroke Dist.**, use the value that we found after measuring the Punch travel distance and subtracting the Blank thickness. The value should be approximately 40.9. After entering the values, click **Yes** and a new Velocity vs. Distance motion curve will be created and displayed (as shown in the following illustration).

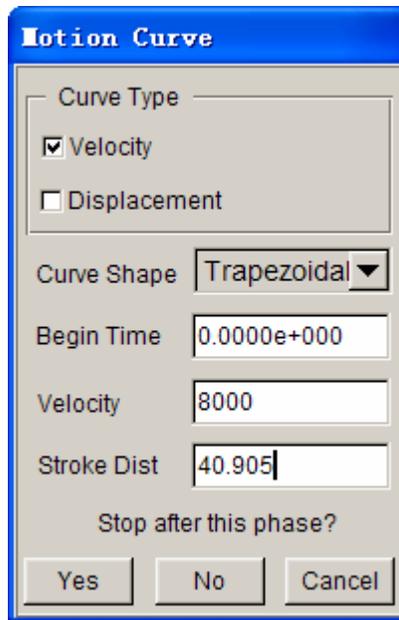


Figure 135: Motion curve dialog box

- Verify that the load curve is identical to the graph as shown on next page.

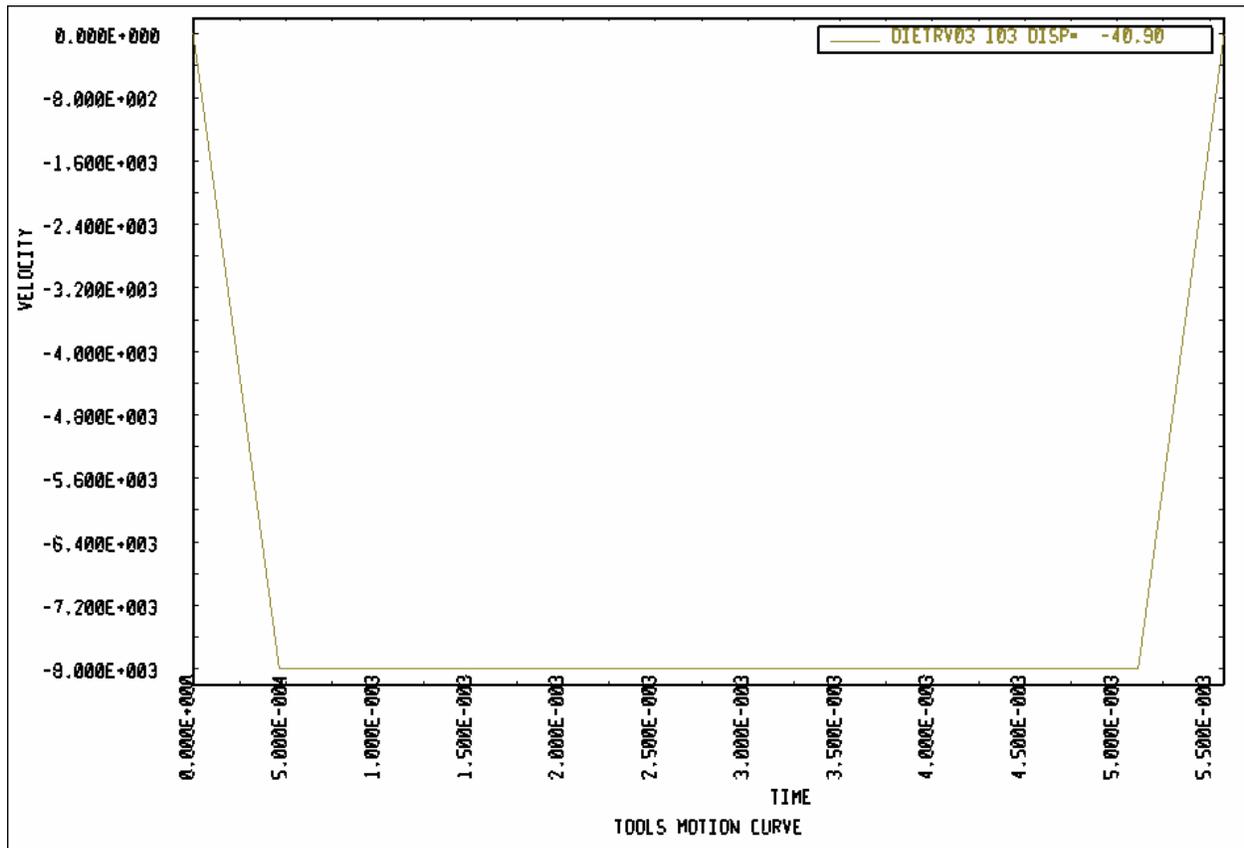


Figure 136: Plotting motion curve

6. Select **Ok** in the **Curves** dialogue to return to the **Tool Load Curve** dialogue. From here, click **Ok** once more to return to the **Define Tools** dialogue. Do not close this dialogue, the next step will start from here.

## XI. DEFINING THE BINDER (LOWRING) FORCE CURVE

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

1. From the **Define Tools** dialogue, select **Binder** from the **Tool Name** menu and select the **Define Load Curve** button.

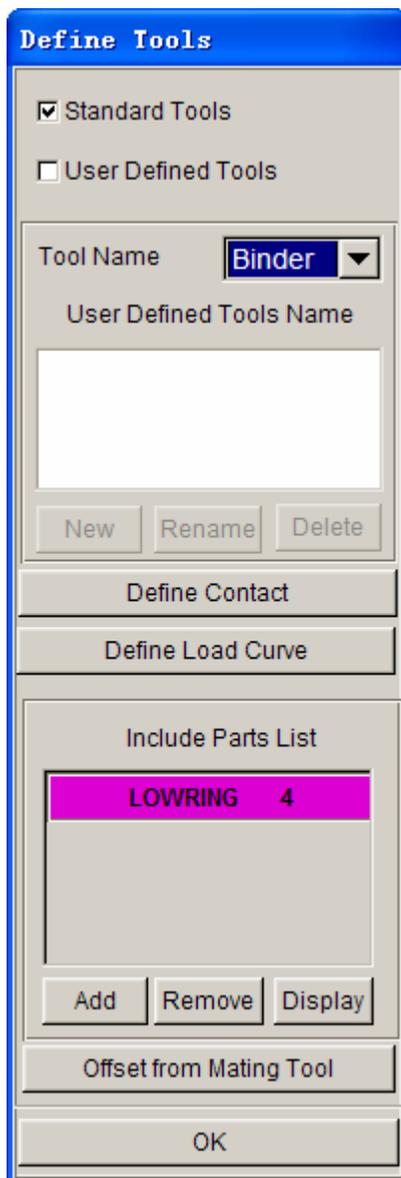


Figure 137: Define tools dialog box

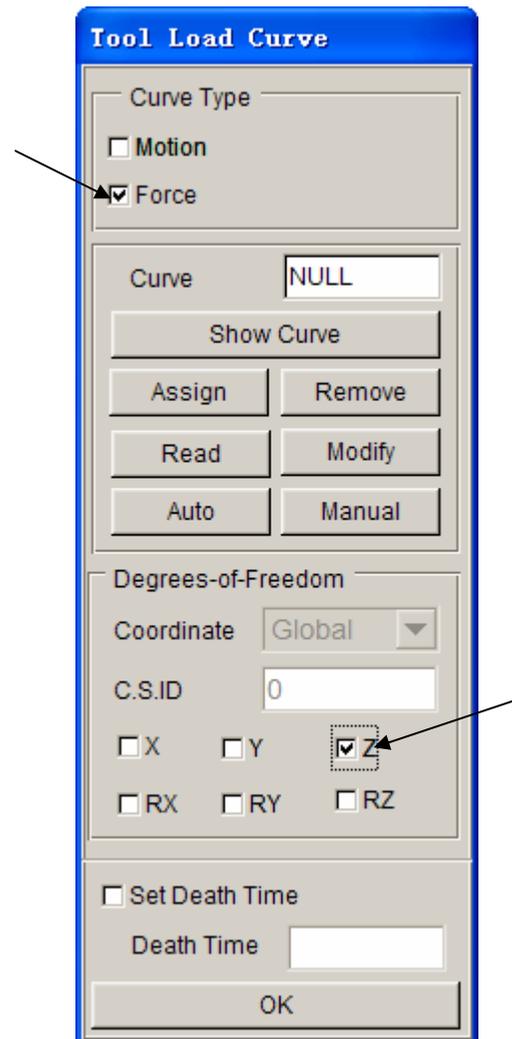


Figure 138: Tool load curve dialog box

2. Select **Curve Type (Force)**. Make sure to select **Z** to define the moving direction and then click **Auto** button.
3. Enter 200000 (N) in the **FORCE** and close the window by clicking **OK**.

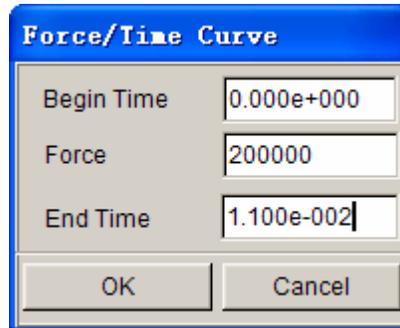


Figure 139: Force/time curve dialog box

4. The **Lower Ring force** curve will be displayed. Verify that it is identical to the curve shown on next page.

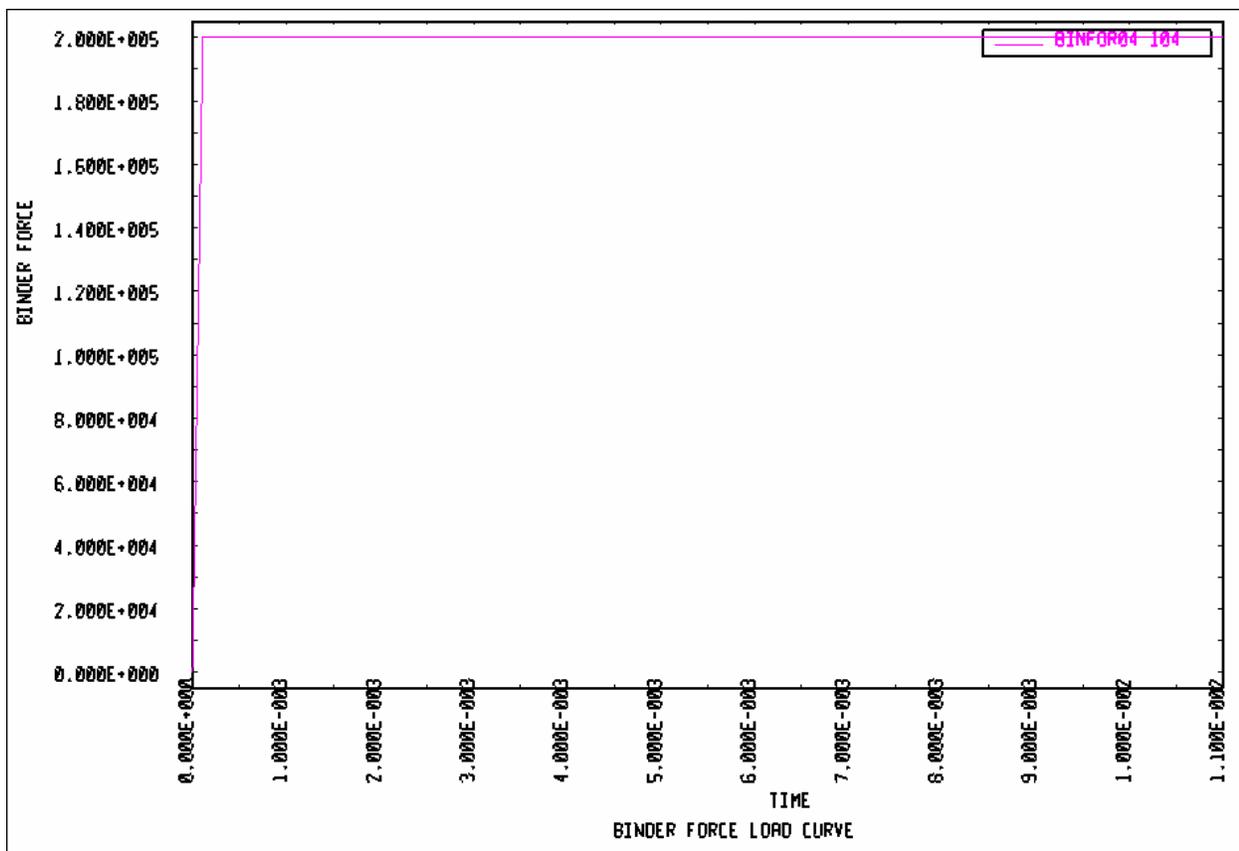


Figure 140: Plotting binder force

5. Select **OK** in the **Curves** dialogue to return to the **Tool Load Curve** dialogue. From here, select **OK** once more to return to the **Define Tools** dialogue.
6. Click **OK** in the **Define Tools** dialogue; we are done defining the motion curves for the tools.
7. Save the changes.

## XII. 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 assigned.

1. Click **Tools**→**Animate** on **Menu bar**.

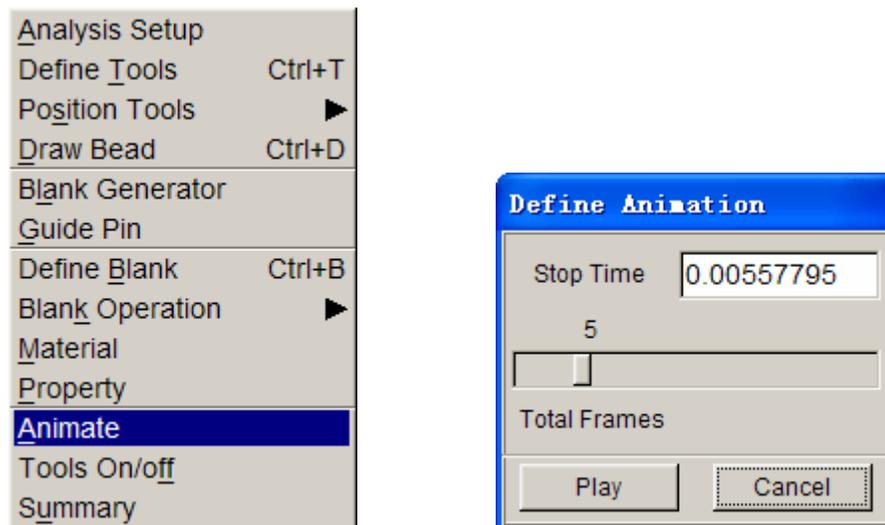


Figure 141: Define animation dialog box

2. Click **Play** to animate the tooling using the default parameters.

*Note:* Depending on the speed of the machine that is used to run eta/DYNAFORM, 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. 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. Also, notice the **Individual Frames** switch on the dialogue. After viewing the animation, click the **Stop** button on the **Animate** dialogue to exit the function.

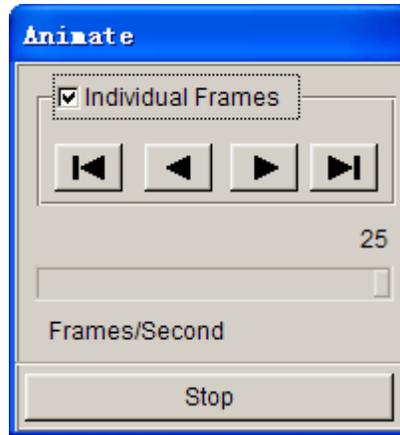


Figure 142: Animate dialog box

4. Save the database.

### XIII. RUNNING THE ANALYSIS

After verifying 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 the user to obtain more accurate results by refining the blank mesh as needed during the simulation. In other words, as the die deforms the blank, areas that demand a finer mesh to capture the tooling geometry will divide to create finer and smaller elements.

1. Select **Analysis**⇒**LS-DYNA...** from the **Menu Bar**.



Figure 143: Analysis menu

2. Click the **Control Parameters** button in the **Analysis Parameters** dialogue.
3. As a new user, it is recommended to use the default control parameters (for more information on them, please refer to the *LS-DYNA User's Manual*). Click **OK**.

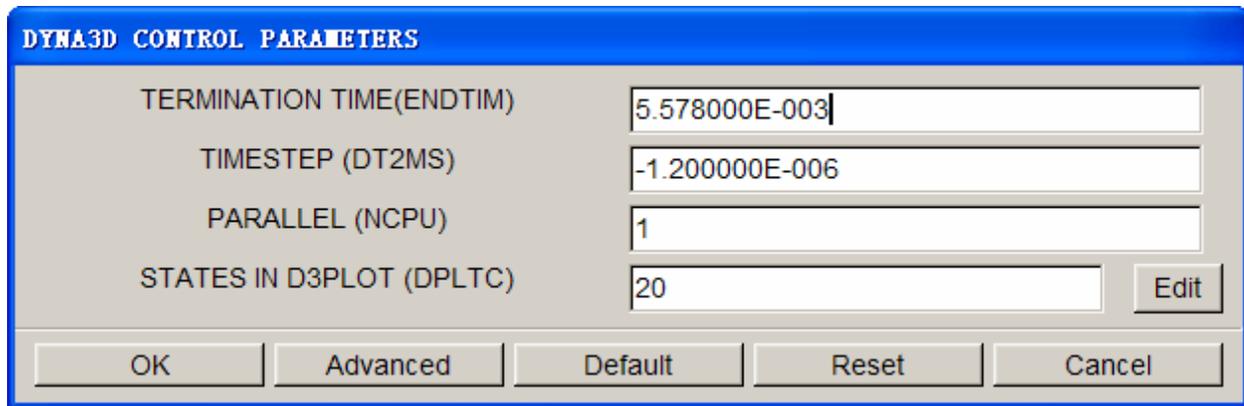


Figure 144: DYNA3D control parameters dialog box

4. There are two methods to run the job.
  1. Select **Full Run Dyna** and press **OK**. The solver will now be running in the background.
  2. For manual submission of the input deck:
    - a. Select **LS-Dyna Input File** and enter a file name (e.g. training) and press **Ok**.
    - b. Find the directory that includes your current example file (training.df), you will find two files “training.dyn” and “training.mod”. All files generated by either eta/DYNAFORM or LS-DYNA will be placed in the directory in which the eta/DYNAFORM database has been saved. This includes all input decks and post processing files.
    - c. Then select **File->Submit Dyna from Input Deck**, find the “training.dyn”, then press **OK**.
    - d. Give a right memory, press **OK** to start the solver.

The solver displays a DOS window showing the status of the job. You will notice that an estimated completion time is given. This time is not accurate since we are using adaptive mesh and the model will re-mesh several times. Also, it is influenced the speed and number of CPU of the machine. However, it does give you a general idea.

5. Once the solver has given you the preliminary estimated time, you can refresh this estimate by pressing **Ctrl-C**. This will momentarily pause the solver which will prompt you to “**enter sense switch:**”. Type the switch command 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.

```

c:\ D:\eta\eta\SOLVER~1\lsdyna.exe

input of data is completed

initialization completed
  476 t 5.5790E-04 dt 1.08E-06 write adaptivity stress file
  476 t 5.5790E-04 dt 1.08E-06 add shell elements from 2780 to 2900

memory needed for solution= 819886

input of data is completed

initialization completed
  504 t 5.8706E-04 dt 1.08E-06 write d3plot file
  605 t 6.9722E-04 dt 1.08E-06 write adaptivity stress file
  605 t 6.9722E-04 dt 1.08E-06 add shell elements from 2900 to 2909

memory needed for solution= 822331

input of data is completed

initialization completed

.enter sense switch:sw2

```

Figure 145: Sense switch

When you submit a job from eta/DYNAFORM, an input deck is created which the solver, LS-DYNA, uses to process the analysis. The default input deck names are **yourdatabase name.dyn** and **yourdatabasename.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*.

*Note: All files generated by either eta/DYNAFORM or LS-DYNA will be placed in the directory in which the eta/DYNAFORM database has been saved. It is necessary to make sure the fold doesn't include other resultant files before running the current job in case part of files are overwritten.*

# POST PROCESSING (with eta/POST)

The eta/POST reads and processes 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.).

## I. Reading the Results File into the Post Processor

To execute eta/POST, click **PostProcess** from **eta/DYNAFORM** menu bar illustrated in Figure 146. The default path for **eta/POST** is C:\Program Files\Dynaform 5.5. In this directory, you can double click on the executable file, **EtaPostProcessor.exe** to open eta/POST GUI illustrated in Figure 147. The **eta/POST** can also be accessed from the programs listing under the start menu of **DYNAFORM 5.5**.



Figure 146: Post process menu

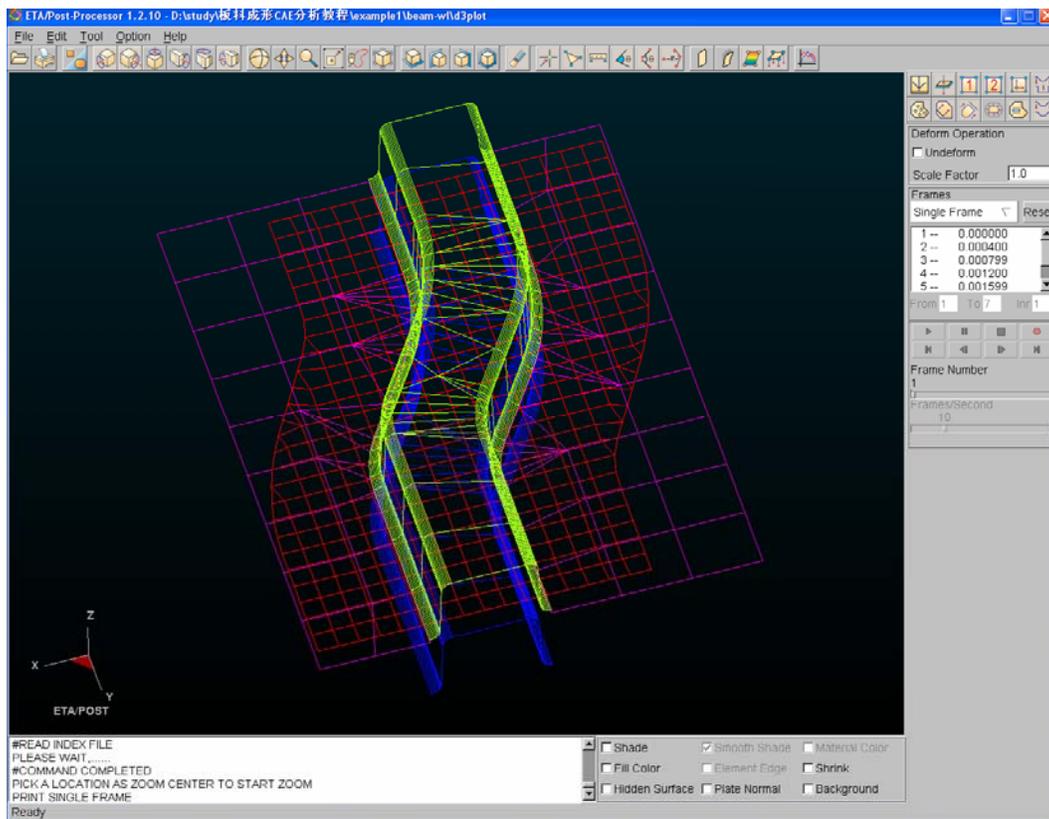


Figure 147: eta/POST GUI

1. From the **File** menu of **eta/POST**, select **Open** function. The Open File dialog box illustrated in Figure 148 is displayed.

- The default **File Type** is **LS-DYNA Post (d3plot, d3drif, dynain)**. This option enables you to read in the **d3plot**, **d3drif** or **dynain** file. The **d3plot** is output from forming simulation, such as drawing, binder wrap and springback, while **d3drif** is generated during gravity loading simulation. The **dynain** is generated at the end of each simulation. This file contains the deformed blank information.

*Note: Please refer to eta/POST User's Manual for description about other file types.*

- Choose the directory of the result files. Pick the **metalforming\_d3plot** file illustrated in Figure 148, and press **Open** button.
- The d3plot file is now completely read in. You are ready to process the results using the Special icon bar, as shown Figure149.

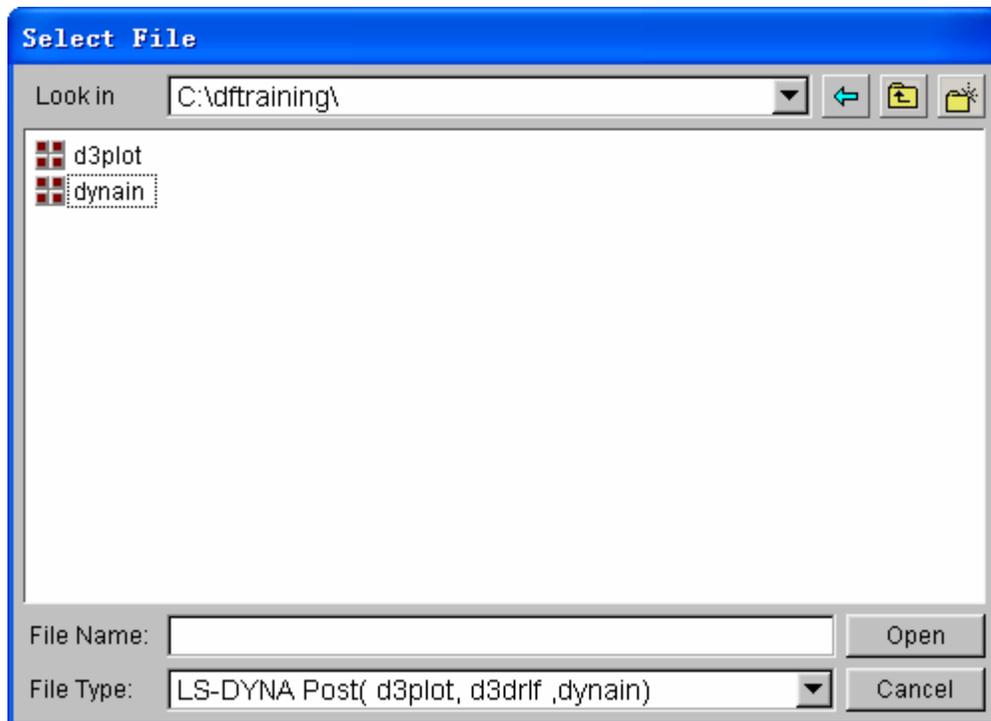


Figure 148: Open file dialog box



Figure149: Special icon bar for forming analysis

## II. Animating Deformation

1. The default **Plot State** is **Deformation**. In the **Frame** dialog, select **All Frames** and press **Play** button to animate the results. See Figure 150.

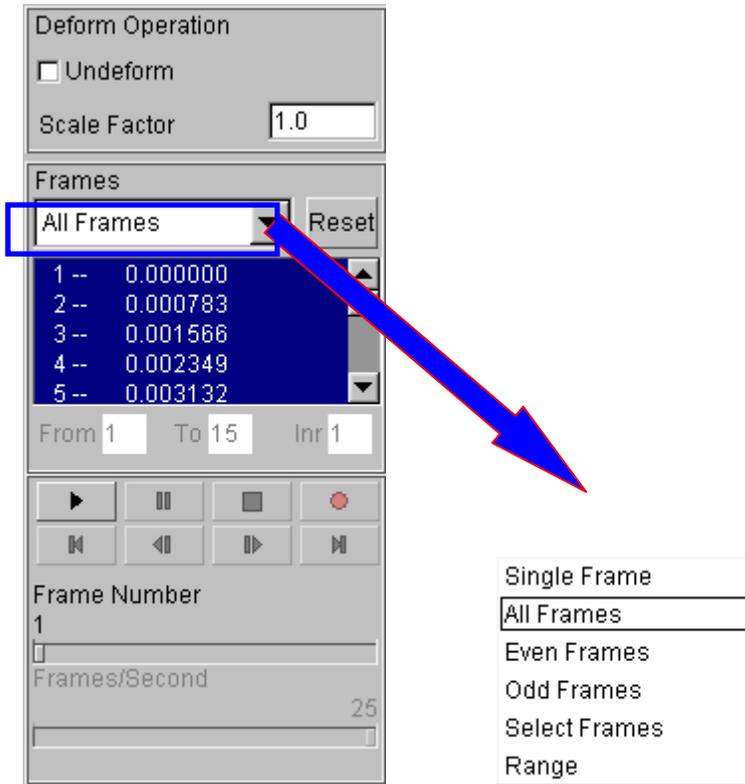


Figure 150: Animating deformation dialog box

2. Toggle on the **Shade** checkbox in the display options illustrated in Figure 151. The **Smooth Shade** option is also toggled on by default to enable smooth and continuous model display.

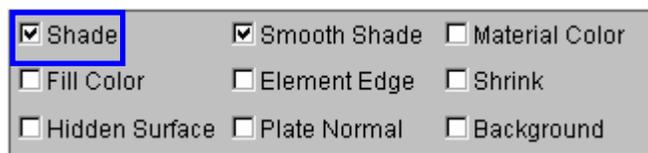


Figure 151: Display options

3. Since it is difficult to see the Blank with all of the other tools displayed, you can hide all of the tools by pressing the  icon from **Icon bar**.
4. In Part Operation dialog, use your mouse cursor to click on all the parts, excluding BLANK, as illustrated in Figure 152. All the tools are hidden from the display area. Then, press **Exit** button to dismiss the dialog box.

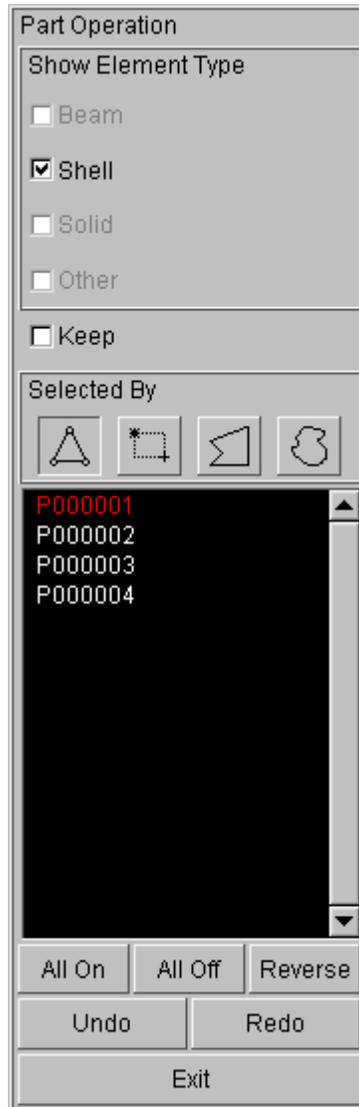


Figure 152: Part turn on/off dialog box

5. You can also change the displayed model using the view manipulation icons on the **Icon bar** illustrated in Figure 153.



Figure 153: Icon bar

### III. Animating Deformation, Thickness and FLD

In eta/POST, you can animate deformation, thickness, FLD and various strain/stress distribution of the blank. Refer to the following examples for animation.

#### Thickness/Thinning

1. Select  icon from the Special icon bar.
2. Select **Current Component** from combo box, either THICKNESS or THINNING. The thickness contour of deformed dome is illustrated in Figure 154.
3. Click **Play** button to animate the thickness contour.
4. Use your mouse cursor to move the slider to set the desired frame speed.
5. Click **Stop** button to stop the animation.

S-BEAM

STEP 17 TIME: 0.011001

COMPONENT: Thickness

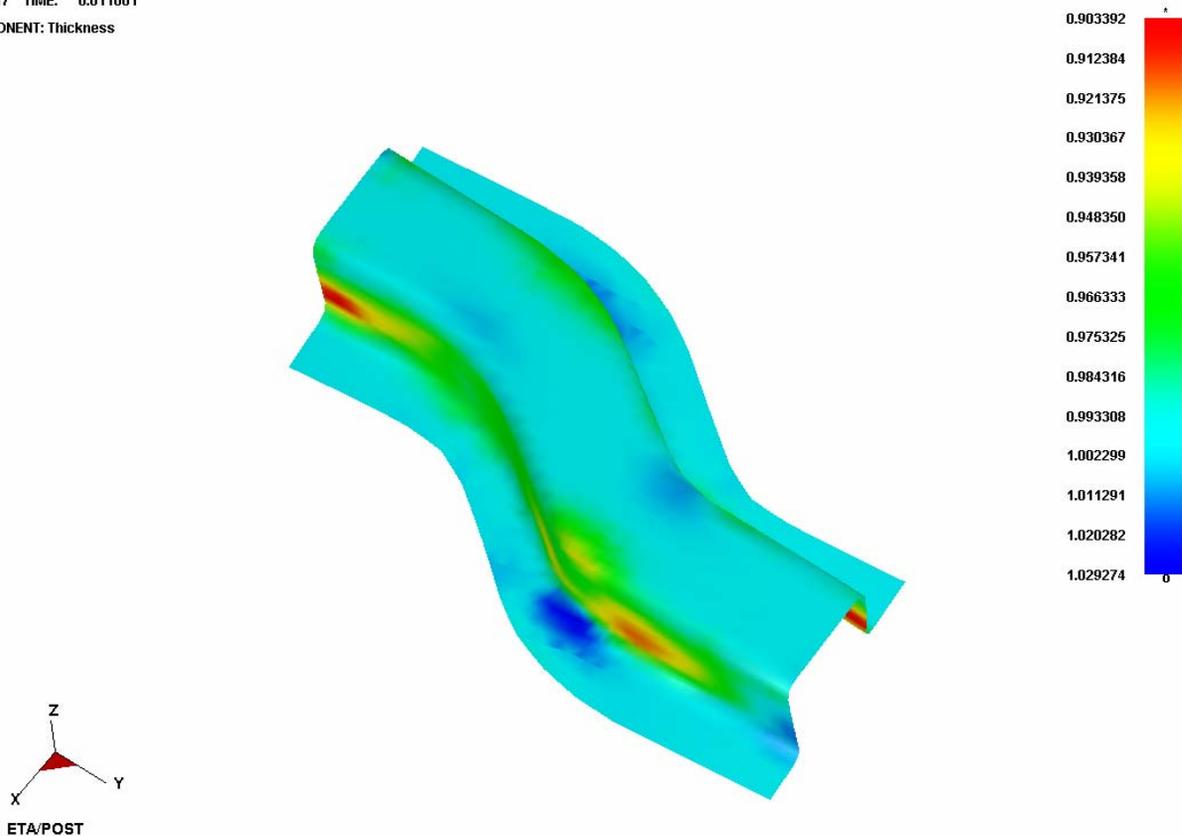


Figure 154: Thickness/Thinning contour

**FLD**

1. Pick  from the Special icon bar.
2. Select **Middle** from the **Current Component** list, as illustrated in Figure 155. The FLD contour of deformed dome is illustrated in Figure 156.
3. Set FLD parameters (n, t, r, etc.) by clicking on the **FLD Curve Option** button to display the FLD CURVE AND OPTION dialog box.
4. Click **OK** button to dismiss the FLD CURVE AND OPTION dialog box.

5. Select **Edit FLD Window** function to define location of FLD plot on the display window.
6. Click **Play** button to animate the FLD contour of deformed dome.
7. Click **Stop** button.

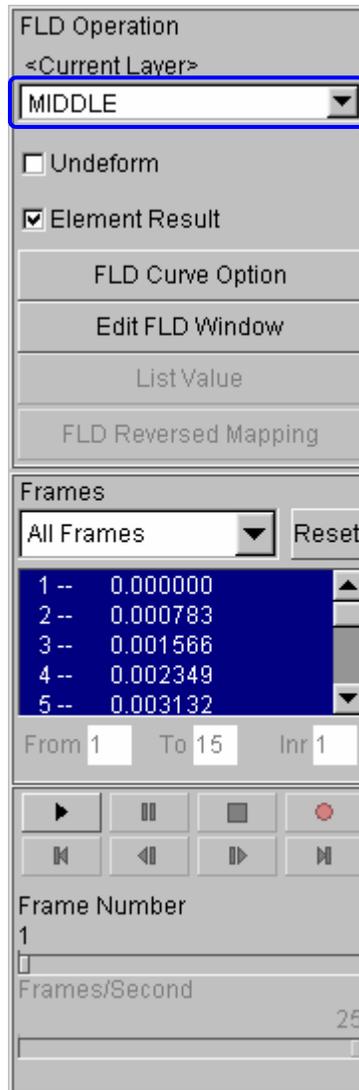


Figure 155: FLD display dialog box

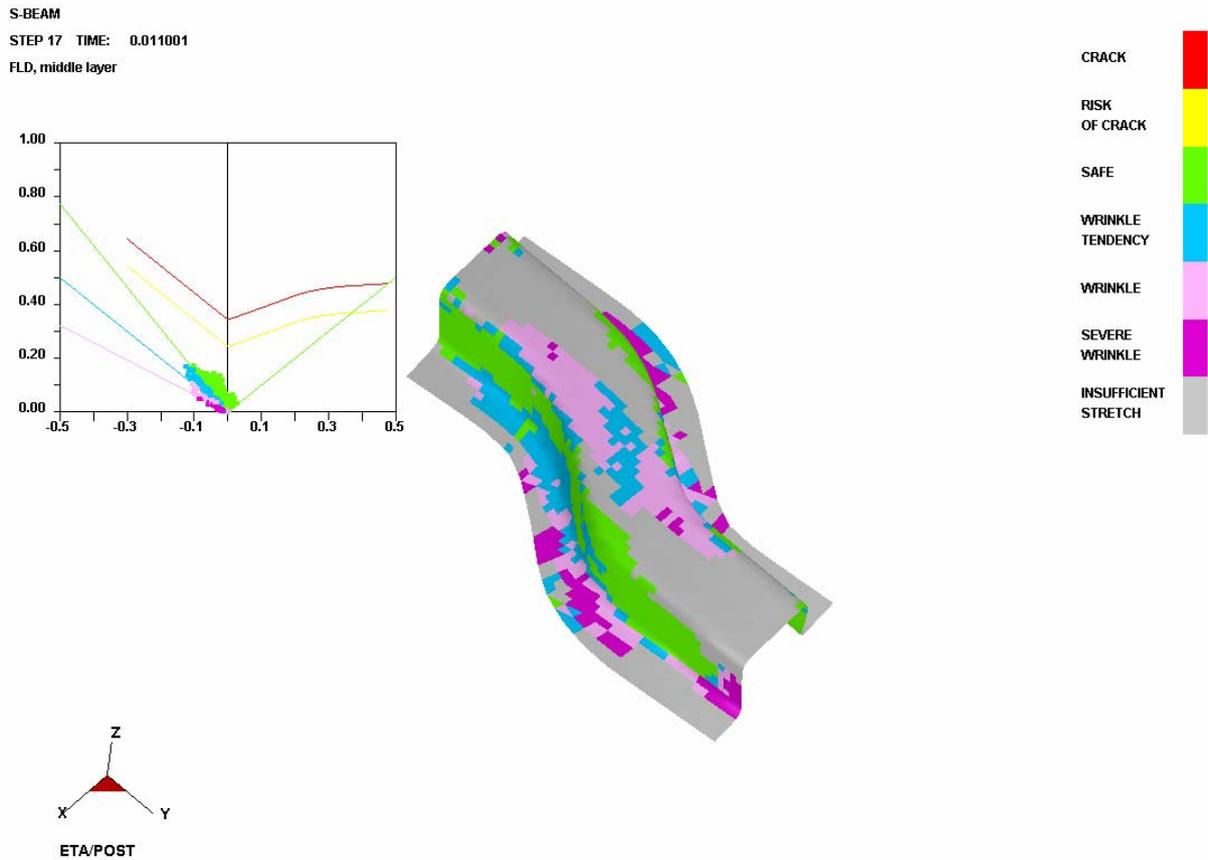


Figure 156: FLD contour

#### IV. Plotting Single Frames

Sometimes, it is much convenient to analyze the result by viewing single frames rather than the animation. To view single frame, select **Single Frame** option from the **Frames** combo box illustrated in Figure 157. Then, use your mouse cursor to select the desired frame from the frame list. You can also drag the slider of frame number to select the frame accordingly.

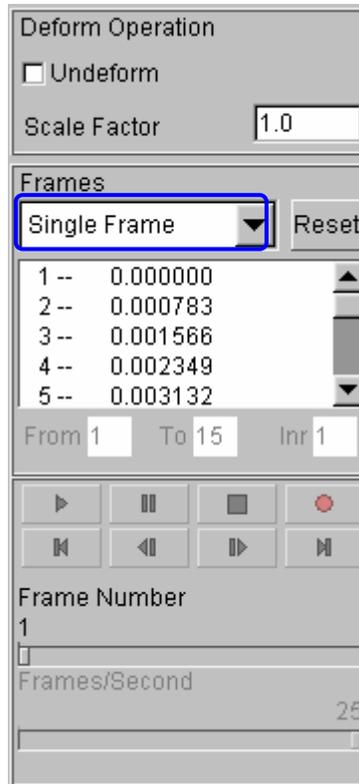


Figure 157: Display of single frame

## V. Writing an AVI and E3D File

eta/POST provides a very useful tool that allows you to automatically create an **AVI** movie and/or **E3D** files via an animation of screen capture. The **Record** button illustrated in Figure 158 is commonly utilized to generate AVI movie and/or E3D files. This is the last function covered in this application example.

### AVI movie

The following procedure is used to generate an AVI movie file.

1. Start a new animation using the procedure provided in Section III.
2. Display the model in isometric view.
3. Click the **Record** button.
4. The **Select File** dialog is displayed.
5. Enter a name of the **AVI** movie file (e.g. traincase.avi) at the input data field of File Name illustrated in Figure 159.
6. Then, click **Save** button.
7. From the dialog box illustrated in Figure 160, select **Microsoft Video 1** from the **Compressor** list and click **Ok** button.

8. eta/POST will take a screen capture of the animation and write the output.

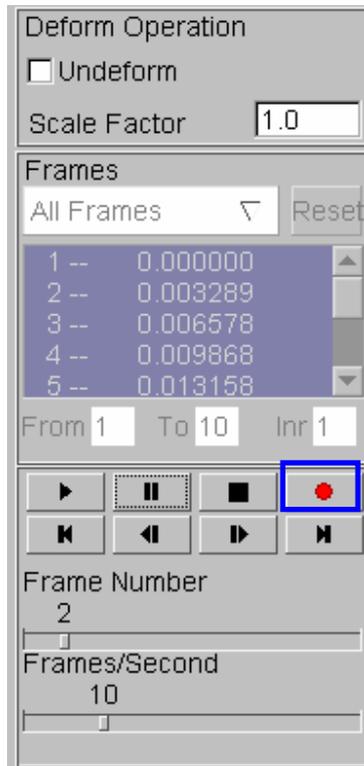


Figure 158: AVI dialog box

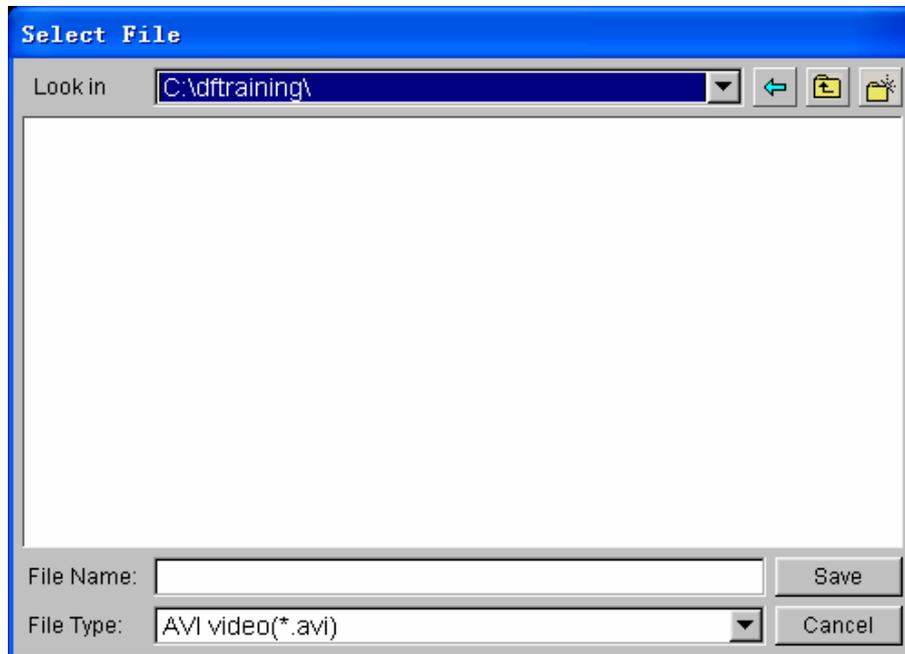


Figure 159: Save AVI file

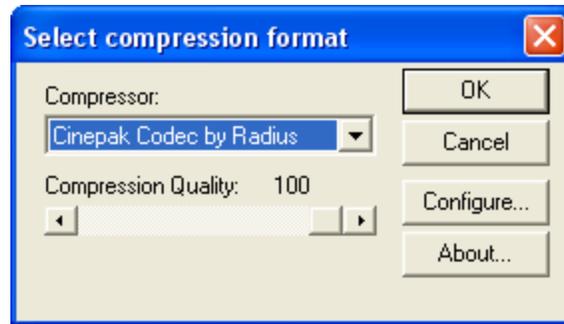


Figure 160: Select compression format

### E3D file

You can also save simulation results in a much compact file format (\*.e3d). The \*.e3d file can be viewed using **eta/3DPlayer** which is provided as a free software to any users.

To create an E3D file, refer to the steps listed below:

1. Start a new animation (thickness, FLD, etc) using the procedure provided in Section III.
2. Display the model in isometric view.
3. Click the **Record** button.
4. The **Select File** dialog box is displayed.
5. Click the drop-down button of File Type to select **E3D Player file (\*.e3d)**.
6. Enter a name of the **E3D** file (e.g. traincase.e3d) at the input data field of File Name.
7. Then, click **Save** button.
8. eta/POST will take a screen capture of the animation and write the output into \*.e3d format.

You can view 3D simulation results using the player. To start the player, select Start→All Programs→Dynaform 5.5→Eta3DPlayer.

## MORE ABOUT DYNAFORM 5.5

Inside the eta/DYNAFORM 5.5 installation directory, there is a file called **.DynaformDefault**. Many key default parameters are included in this file. Advanced users can customize these default parameters.

For Unix/Linux users, the **.DynaformDefault** file is located under both the installation directory and the user's home directory. The **.DynaformDefault** file located under the installation directory will take precedence over the one located in the user home directory.

## CONCLUSION

This concludes the training guide's basic overview of **eta/DYNAFORM 5.5**. This manual is meant to give the user a basic understanding of finite element modeling for forming analysis, as well as displaying the forming results. It is by no means an exhaustive study of the simulation techniques and capabilities of **eta/DYNAFORM**. For more detailed study of **eta/DYNAFORM**, the user is urged to attend a **eta/DYNAFORM** training seminar.

Please refer to the **eta/DYNAFORM** and the *LS-DYNA User's Manuals* for detailed description of individual functions and analysis settings.

eta/DYNAFORM team