eta/DYNAFORM TRAINING MANUAL
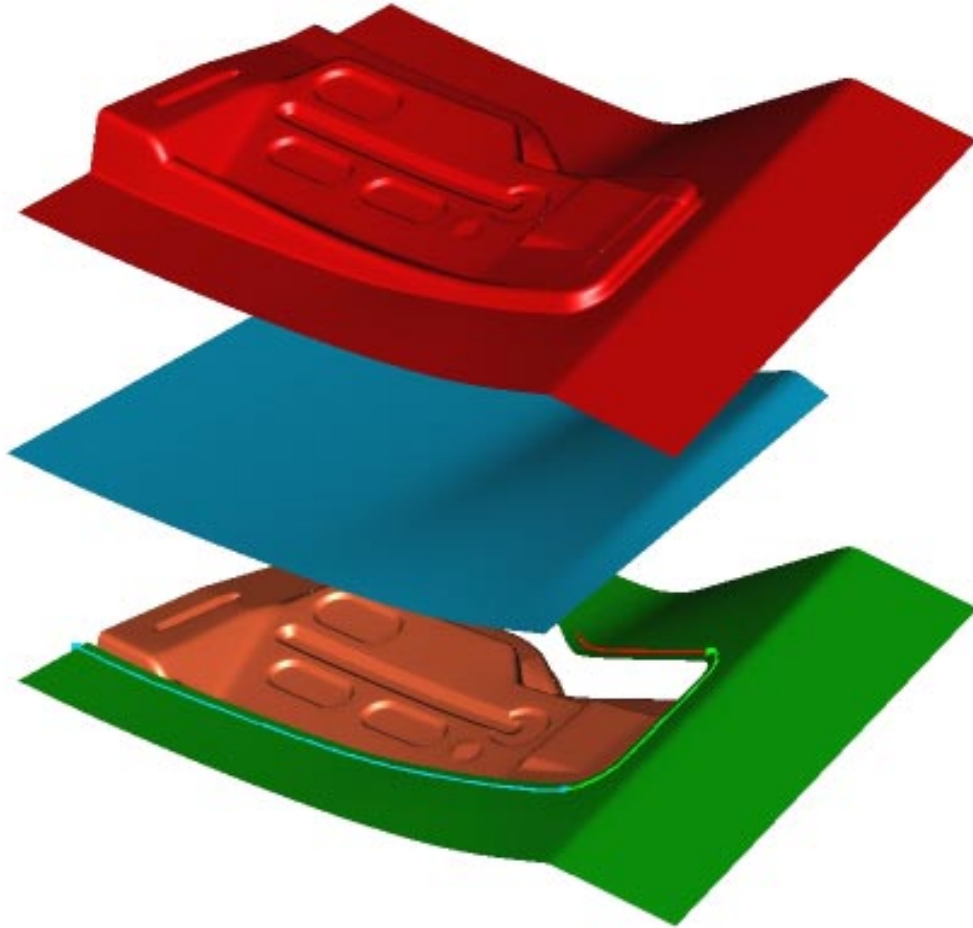
**An LS-DYNA Based Sheet Metal Forming
Simulation Solution Package**

Version: 3.0
Release Date: May 1, 1999

eta/DYNAFORM
Training Manual

Version: 3.0

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 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.

TABLE OF CONTENTS

eta/DYNAFORM TRAINING MANUAL

This training manual was created to familiarize new users with eta/DYNAFORM Version 3.0. The exercises in this training manual include pre-processing line/surface data, creating a finite element model, defining parts as tools, setting forming parameters (blank material and property, punch velocity, binder force, etc.), regulating the values of adaptive mesh, and seamless and multi-stage sheet metal forming processes.

It also includes creating an LS-DYNA input deck file and post-processing the results of LS-DYNA including blank trimming and springback. In advanced training, a more complex drawing case is introduced that examines defining draw beads and adaptive mesh. Multi-stage sheet metal forming processes also are dealt with in this case.

eta/DYNAFORM 3.0 MENU SYSTEM

eta/DYNAFORM 3.0 is a pre- and post-processor which utilizes the MOTIF Graphic User Interface. The main window of the user interface consists of six parts: the Menu Bar, Display Area, Prompt Window, Dialog Window Display, Display Options and Icon Bar.

Menu Bar **Icon Bar**

Dialog Window Display **Display Area** **Prompt Window** **Display Options**
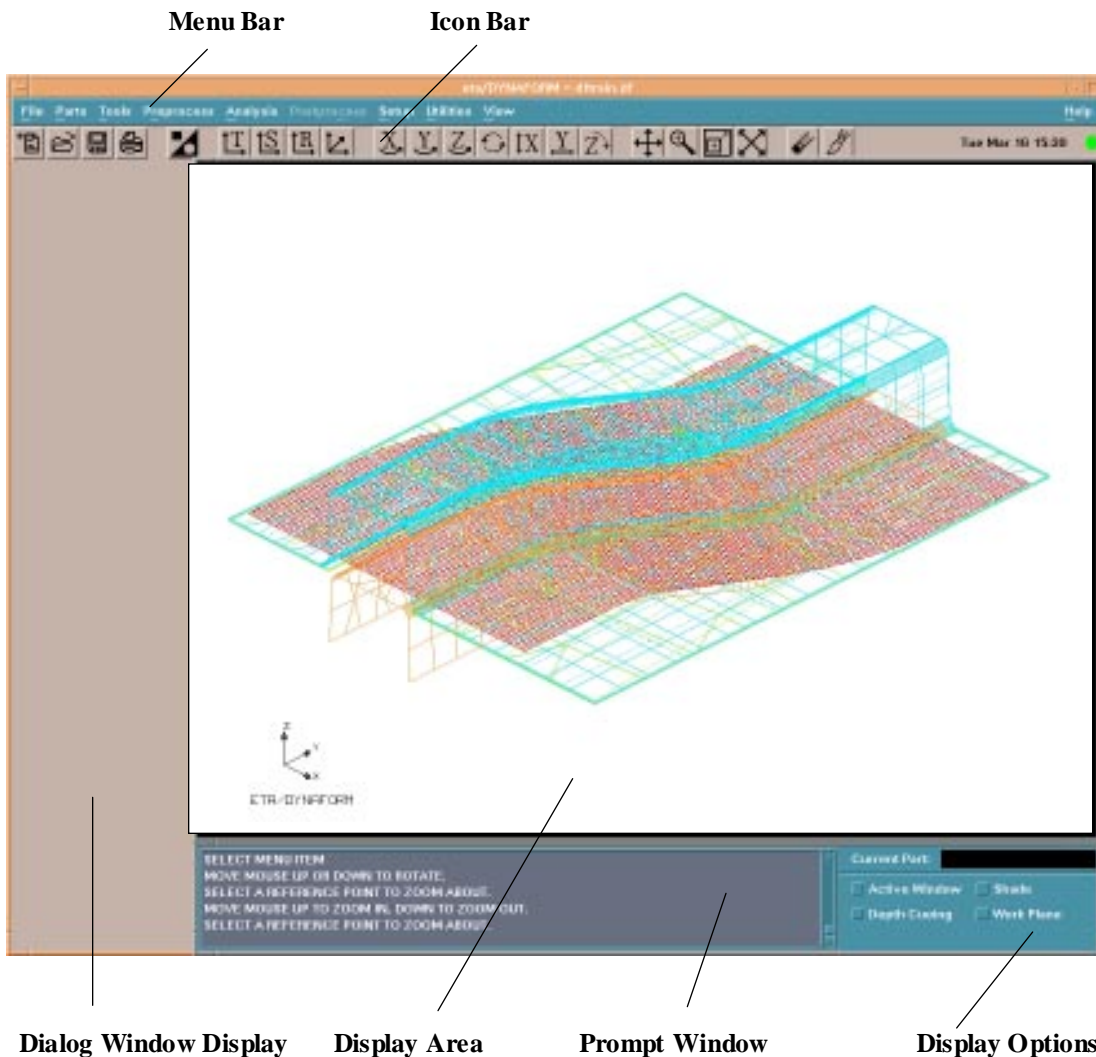
MENU BAR

The top of the eta/DYNAFORM 3.0 interface is composed of a series of menus referred to as the "Menu Bar". The menu system is setup as a tree structure; choosing an option on the menu bar opens a menu that branches out into sub-menus or pop-up dialog boxes. Brief descriptions for these menus follow:

File   1. Open, create and save an eta/DYNAFORM database file.
2. Import and export data to and from eta/DYNAFORM.
3. Print setup and print.

Parts   The parts menu is used to edit, manipulate and define parts. By default, every entity created or read into DYNAFORM is assigned to a part.

Tools   1. Define parts as tool/blank.
2. Create and assign blank materials and properties.
3. Generate and assign material stress-strain relationships, tool travel velocity, and binder force.
4. Calculate the minimum distance between tool pieces, and auto position tool pieces and blanks.
5. Animate tool motion; define draw bead geometry and force load curves.

Preprocess   1. Generate and modify Point/Line/Surface data.
2. Create, copy, transform and manipulate Nodes/Elements.
3. Check element criteria (boundary, overlap, warpage, aspect ratio, etc.).
4. Create boundary condition.

Analysis   Set LS-DYNA control parameters, write out an LS-DYNA input deck file, and run analysis directly from eta/DYNAFORM window.

Postprocess   View analysis results and draw Forming Limit Diagrams.

Setup   Set up the concerned parameters in building a finite element model.

Utilities   Commonly used utilities are grouped in this menu.

View   Manipulate the position and perspective of the model display.

DISPLAY AREA
 Models and graphs are displayed in this window.

PROMPT WINDOW
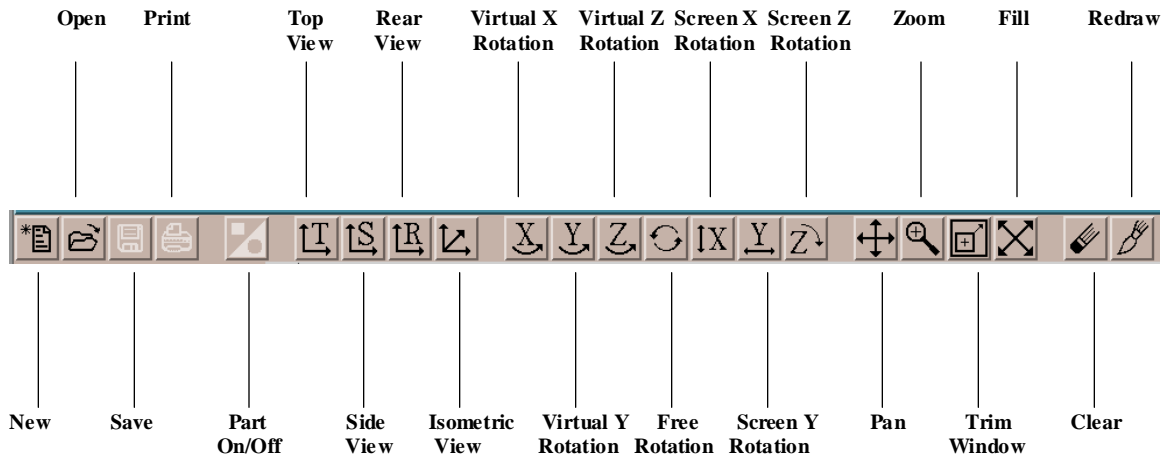 Displays messages and prompts to the user.

DIALOG WINDOW DISPLAY
 Commonly used dialog boxes such as **Create Part** are displayed in this area for user convenience.

DISPLAY OPTIONS
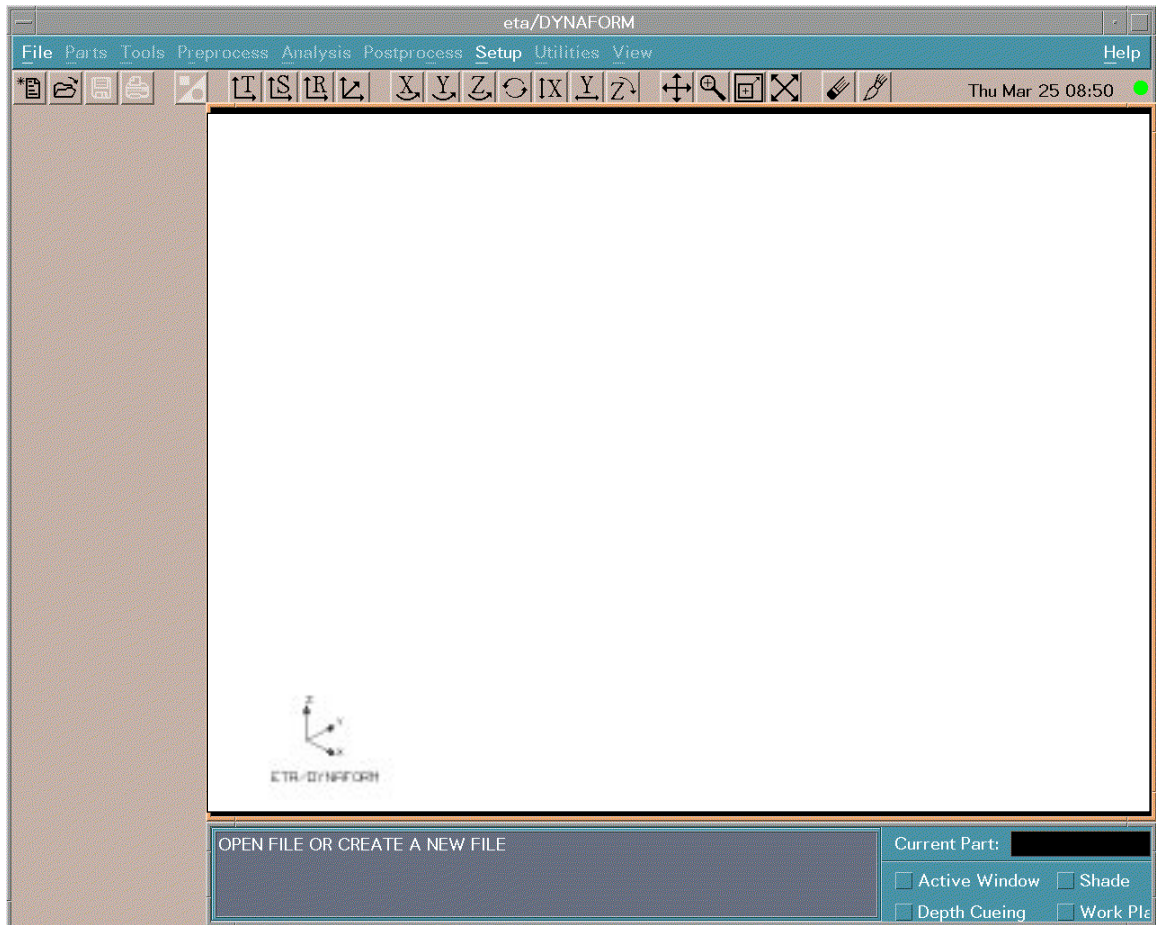Displays the current part and model display properties.

ICON BAR

Refer to the User's Manual for additional information on each icon and its function.

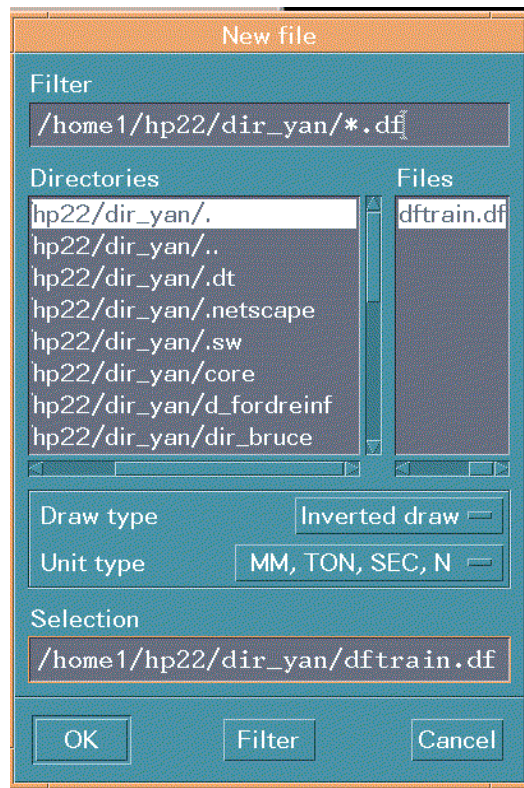# Opening/Creating an eta/DYNAFORM Database File

Getting Started

Type '**dynaform**' (or an alias if one exists) in the UNIX window to activate the eta/DYNAFORM program.

Creating an eta/DYNAFORM Database File

Command: **File/New**

1. Select **New** from the **File** menu and eta/DYNAFORM displays the **New File** dialog box.

2. Select **Inverted Draw** from the **Draw Type** pull-down menu.

3. Select **MM, TON, SEC, N** option in the **Unit Type** pull-down menu.

4. Choose the directory in which you want the eta/DYNAFORM database file to be created. Type the database file name in the type box. For this exercise, type '**dftrain.df**.' The recommended practice is to add the extension **.df** (for dynaform) to newly created files.

5. Click **OK** to exit. The eta/DYNAFORM database file is created and the file name is displayed in the top of the prompt window.

Note: Refer to the Application Manual for a description of draw type.

Reading Geometry Data

Command: **File/Import**

1. From the **File** menu, select **Import** and the **Import Files** dialog box is displayed.
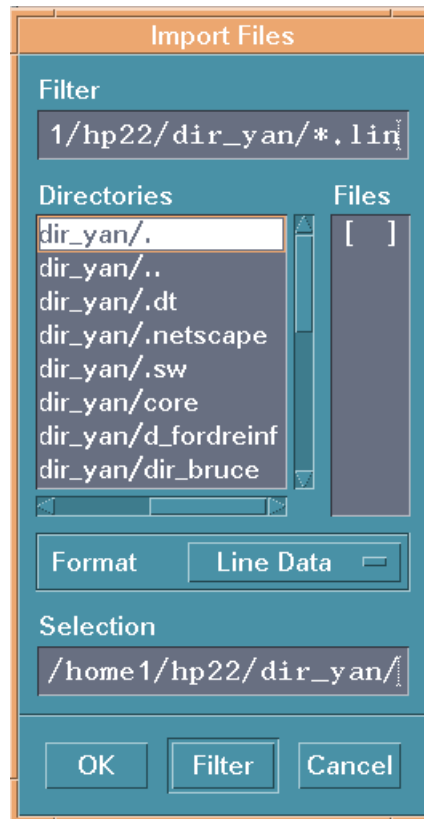
2. Select the **Format** pull-down menu and choose between the two types of geometry data file: IGES and line data. For this exercise, select **Line Data**. All files with the suffix **.lin** will be listed in the **Files** box.

3. From the file list, select BINDER.LIN and click **OK**. The geometry will be displayed in the X-Z plane view.

4. Repeat the above steps and read in the file BLANK.LIN.

   Note: DYNAFORM/FEMB files are line data files created (written) by either eta/DYNAFORM or eta/FEMB. IGES files will have the suffix **.igs**.

Save/Save As

When you finish a session with eta/DYNAFORM, save your model to the eta/DYNAFORM database file before you close it. You should also save periodically while you work so that you do not lose your work in the event of a power interruption or hardware problem.

There are two commands that you can use to save a database file: **File**/**Save** and **File**/**Save As.**

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

2. Use **Save As** to create a new file under a new name.

View Manipulation

At the top of the Display Window, the center of the Icon Bar contains several functions for manipulating the model's orientation. The user may manipulate the view by clicking on any of these options: Top, Side, Rear, Isometric, Free Rotation, Axis Rotation, Zoom, Pan, Fill, Trim Window or Active Window. The user can also open the View menu to get a list of other functions related to model display.

1. Select **Isometric**. This places the displayed geometry in an isometric view.

2. Rotate the geometry dynamically about the Z-axis approximately 90° by using the **Z-Axis** function of **View/Rotation** sub-menu.

3. Select **Fill** from the icon bar. This makes the displayed geometry fill the screen.
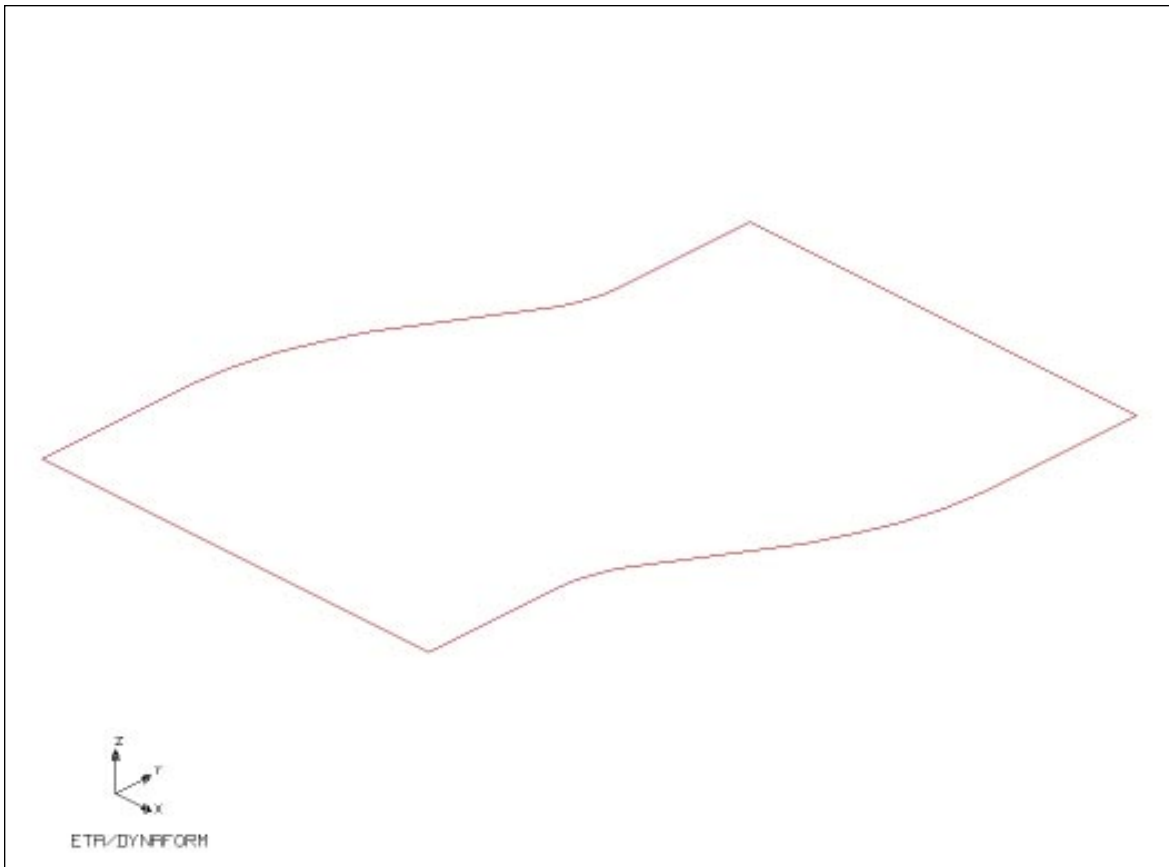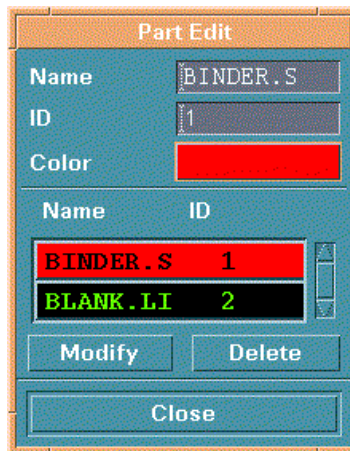
Editing Parts in the Database

All geometry in eta/DYNAFORM is based on parts. Every entity, by default, will be created or read into a part. The parts menu is used to edit, manipulate, and define parts.

Command: **Parts/Edit**

1. From the **Parts** menu, choose the **Edit** button. The **Part Edit** dialog box is displayed with the two parts that are defined in the database.

The parts are listed by part name and property identification number. If the part name is displayed in color, it signifies that the part is currently on. Otherwise, the part name will be displayed in white signifying it is off. The user can modify the name, ID number and color of the parts, create a new part or delete parts.

2. Select **Color**. This displays the **Select Color** dialog box. Select the color yellow to change the color of BINDER.S from red to yellow and select **Modify**. Change the color back to red by repeating the step.

3. Choose **Close** to exit.

Current Part

All lines, surfaces and elements created will automatically be included in the current part. The current part name is displayed in the Display Options window at the lower right of the graphics screen in its part color. Prior to creating any new lines, surfaces and elements, make sure the desired part is current.

When automeshing surfaces, the user may assign the intended elements to each individual part containing surfaces; thus, having the desired target part current is not necessary.

Command: **Part/Current**

1. Select **Current** from the **Part Menu** and the **Part Current** dialog box will display.
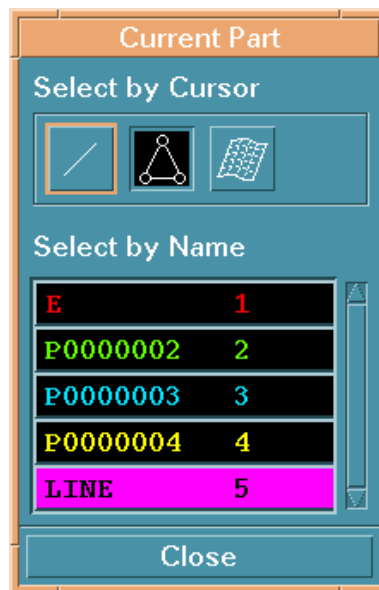

2. The default function is **Select by Cursor at Line**. Click on one of the lines of the BLANK (green part) with the left mouse button. The part BLANK.LI will become the current part. The user can also use **Select by Name** to perform the same command.

3. Select **Close** to exit.

Turning Parts On/Off

Command: **Parts/Turn On**

1. From the **Parts** menu, select **Turn On** and the **Part Turn On/Off** dialog box will be displayed.
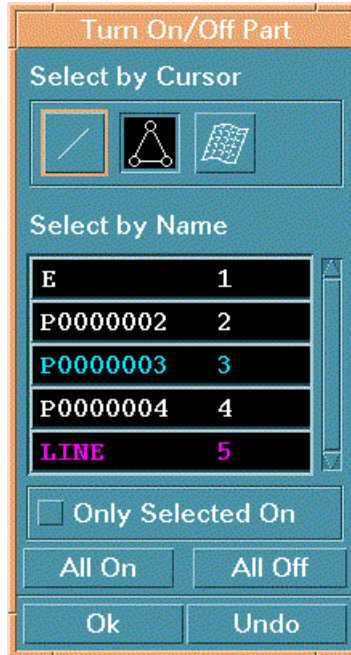

2. Select **Cursor at Surface** and click on BINDER.S (red part). This part will be turned off. The user can do the same thing with **Select by Name**.

3. Select **OK** and proceed with the next command.

4. Select **Fill** from the **Icon Bar.**

Modeling From Line Data
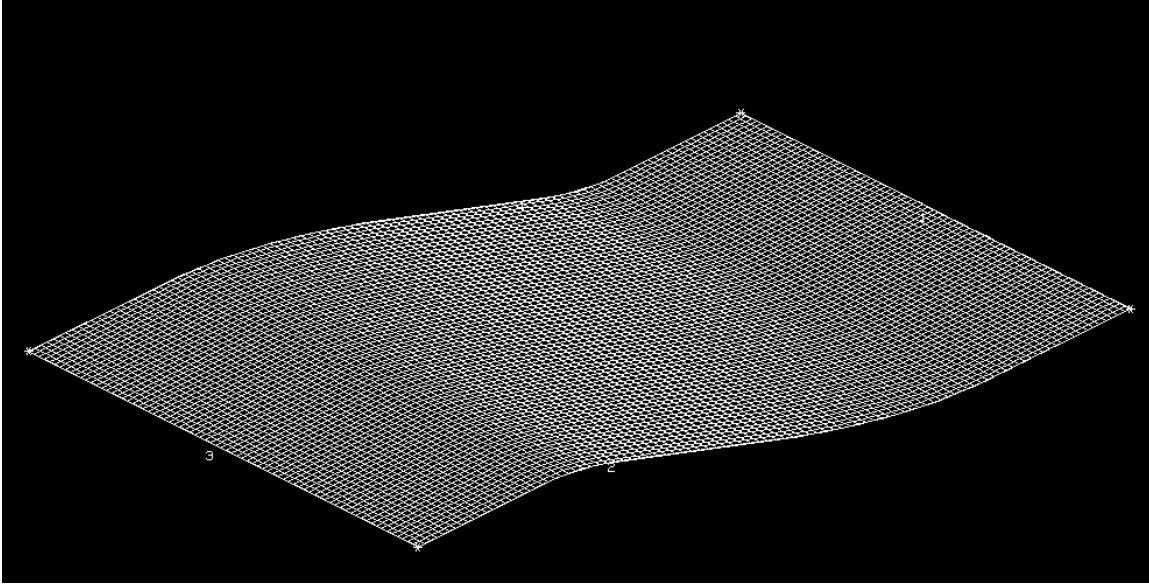
Command: **Preprocess /Element /4 Line Mesh**

1. After selecting the **4 Line Mesh** icon, the **Control Keys** dialog box displays. Choose **Select Lines** and pick four lines of the BLANK sequentially in a clockwise or counter-clockwise order, starting from one of the straight edges.

2. Enter a reasonably fine mesh density of '75' for N1 and '100' for N2 from the **No. of Elements** dialog box and click **OK.**
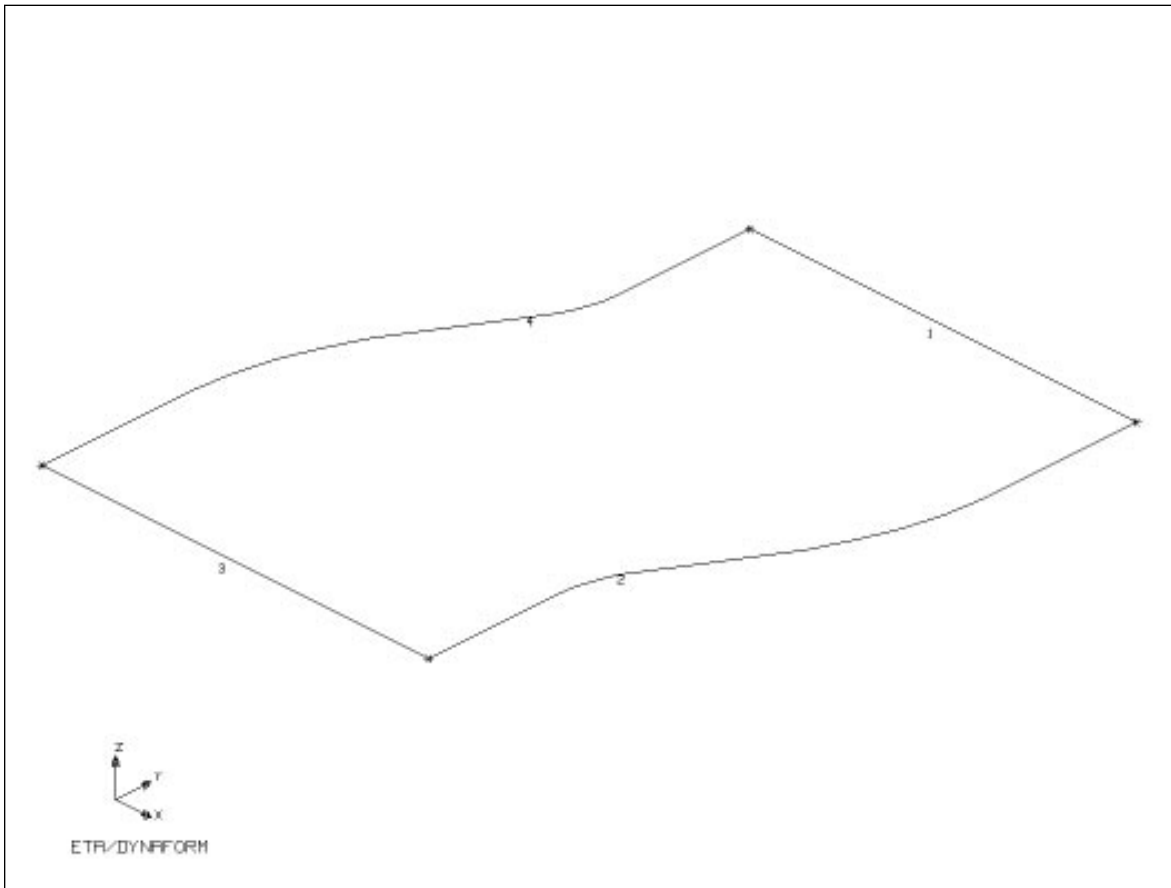
Note: N1, N2, N3, N4 are the number of elements to be created along each of the 4 sides. If the mesh is uniform, only N1 and N2 need to be defined.

3. The generated elements are displayed in white. eta/DYNAFORM prompts:
   > ACCEPT MESH?

4. Click **Yes** with the left mouse button to accept the mesh. This enables the user to see the results of the mesh.

5. Select **Exit** to exit the **4 Line Mesh** function.

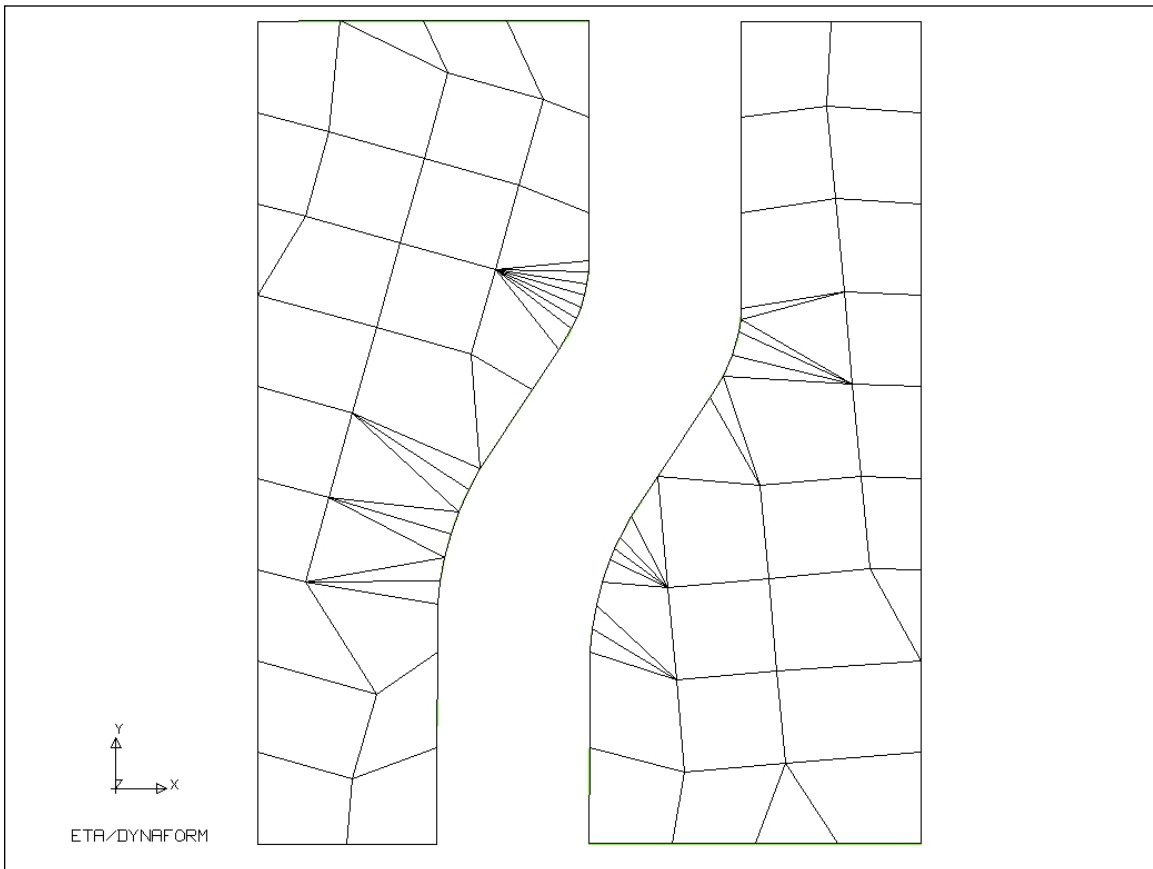6. Select **Save** to save the recent changes.

Auto-Meshing Surface Data

1. Turn the BLANK off and the BINDER on by **Parts/Turn on.**

2. Select **Preprocess/Elements/Surface Mesh** and the **Surface Mesh** dialog box will be displayed.
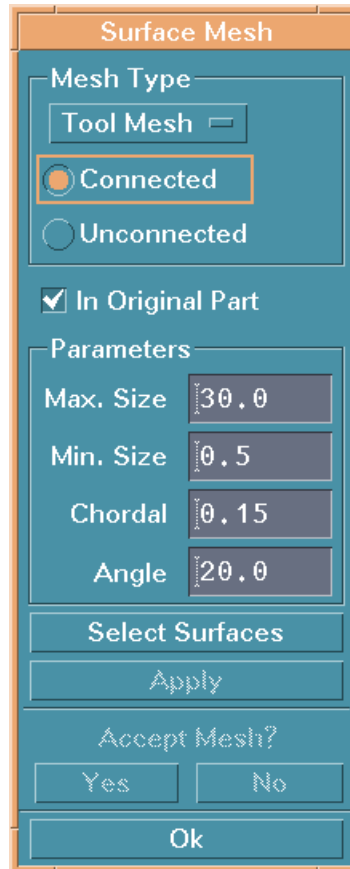
3. In the **Mesh Type** box, select **Connected**.

4. Select **Select Surfaces** and the **Select Surface** dialog box will be displayed. The user can select the surfaces by part, by window or by mouse pick. In this exercise, select **Displayed Surf** and click **OK**.

5. In the **Surface Mesh** dialog box, enter '16' to set the **Maximum element size**. Accept the default values for the **Minimum element size**, **Chordal deviation** and **Angle**. **Chordal deviation** controls the number of elements and **Angle** controls the feature line.

6. Toggle **In Original Part** on. eta/DYNAFORM creates and stores the elements in the part from which they were created. If **In Original Part** is off, eta/DYNAFORM creates and stores the elements in the current part.

7. Select **Apply** to create and optimize the mesh. The generated elements are displayed in white. eta/DYNAFORM prompts:
   > ACCEPT MESH?

8. Select **Yes** to accept the mesh.

9. Select **OK** to exit **Surface Mesh** function.

10. Toggle **Display Surface** off in the **Setup** menu to turn off the surface display.

11. Toggle the surfaces back on and then turn the BINDER off under **Parts/Part on**.

12. Select **Save** to save the change.

Tool Definition

The Binder and Blank are now modeled. The next step is to read in the Die surfaces, auto-mesh, and then create the Punch. This is done by copying the Die elements and offsetting them by the thickness of the Blank. The parts will then be assigned to a respective Tool and most of the analysis parameters will be defined. This is done using the functions in the **Tools** menu.

Auto-Meshing the Die Surface Data

1. Read in the file DIE.LIN under **File/Import** ...

2. Repeat Step 1-12 in **Auto-Meshing Surface Data** to auto-mesh the die surface. Set the maximum element size to '16', the minimum element size to '1' and the chordal deviation ratio to '0.05'.
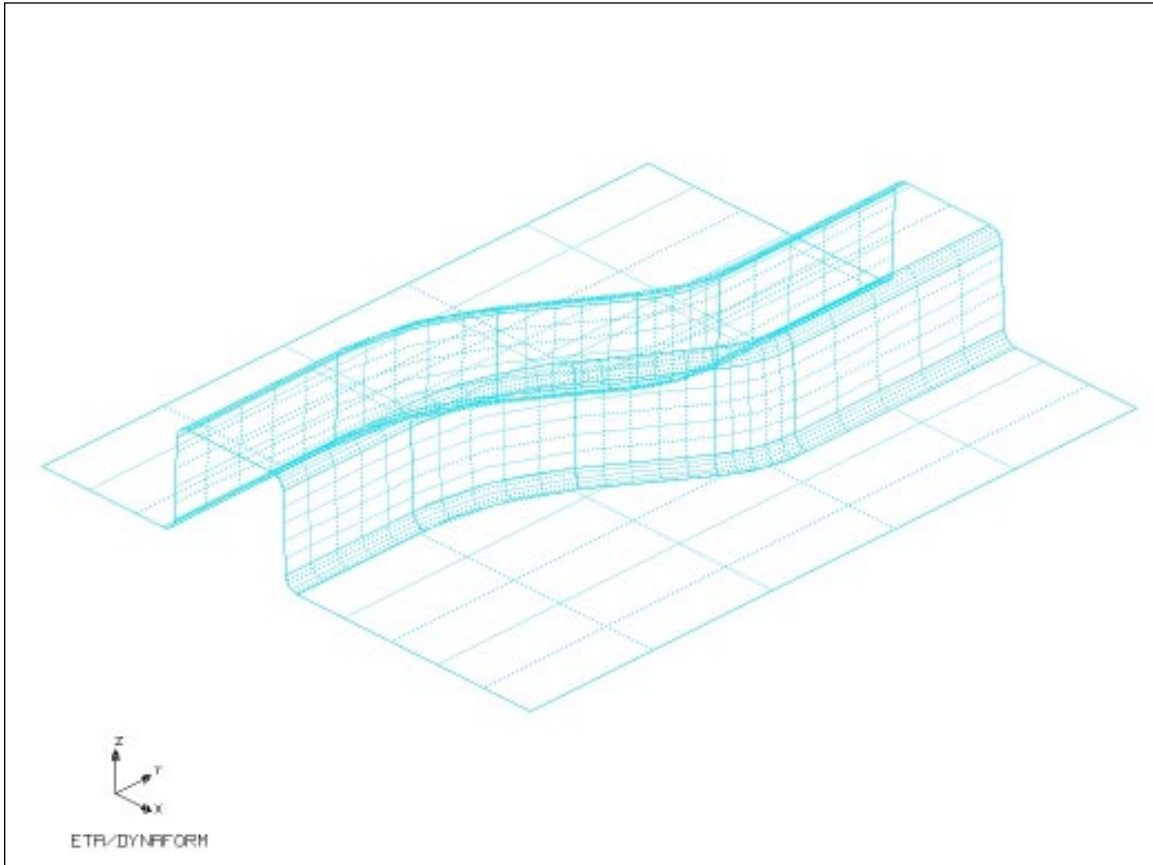
3. Next, select **Preprocess/Model Check/Auto Plate Normal**.
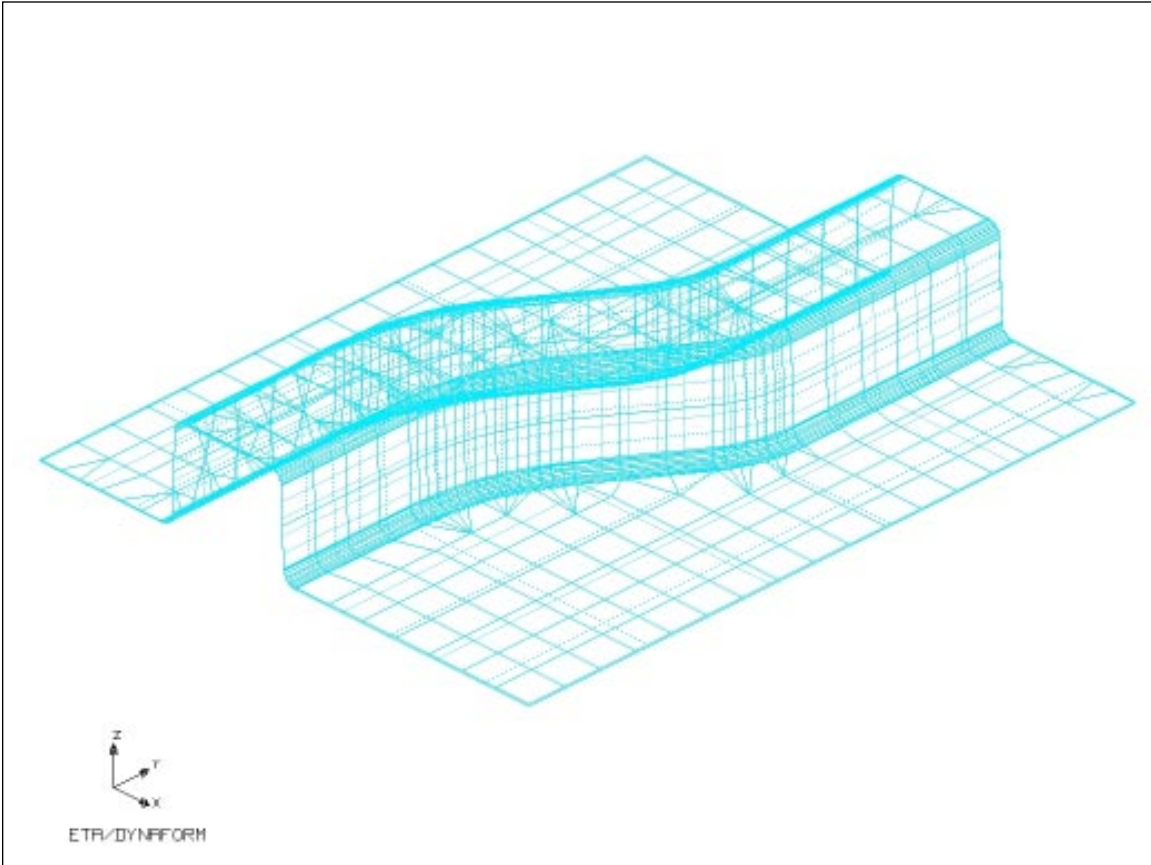
4. Select **All Active Parts** in **Control Keys** dialog box and click **Done.**

5. Pick an element of the Die with the left mouse button.

6. An arrow displaying the normal direction of the element selected will be shown. This arrow should be pointing in the **-**Z direction. If not, select **No** when asked if the direction is acceptable. The element normal will be checked and re-oriented in the -Z direction.
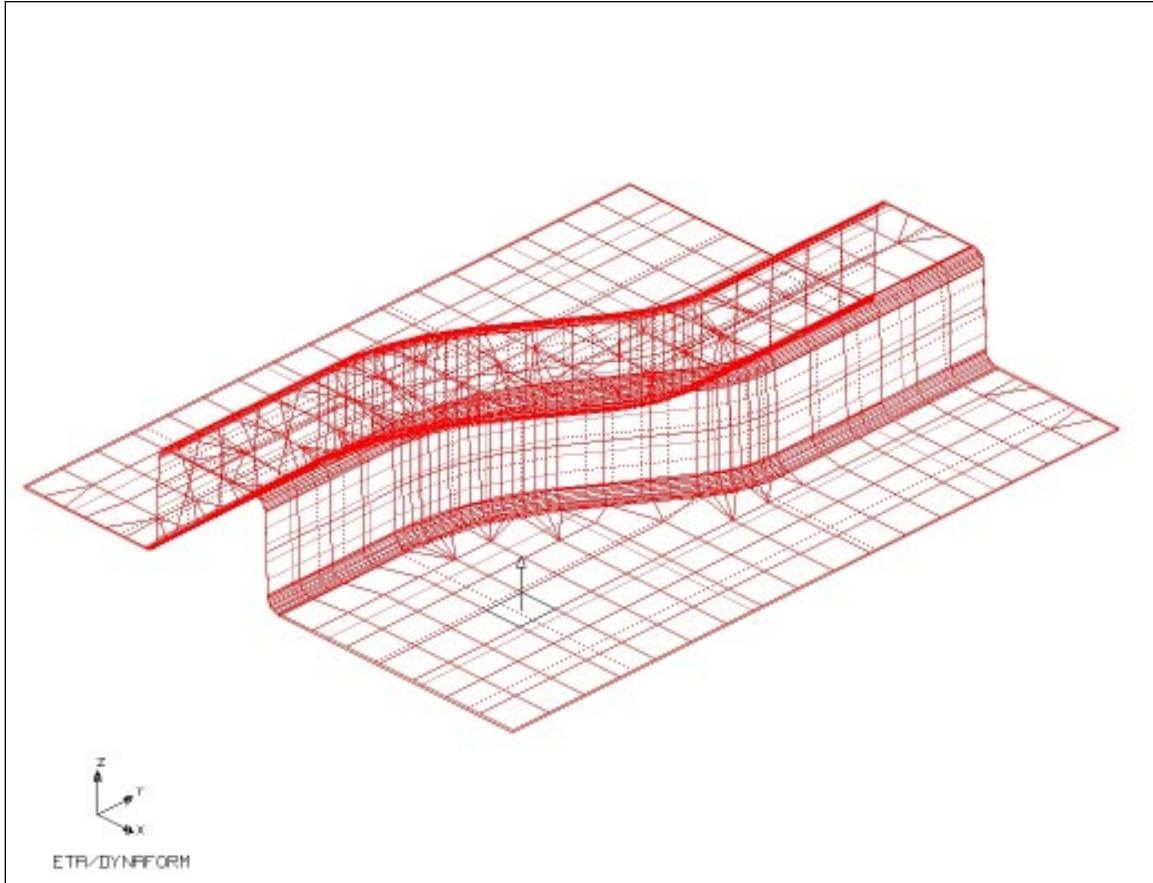
7. Select **Save** to save the change.

Offsetting the Die Mesh to Create the Punch

After the Die elements have been normalized, the next step is to create the Punch by copying the elements of the Die and offsetting them in their normal direction by the thickness of the Blank.

Command: **Parts/Create**

1. Select **Parts/Create** to create a new part called "Punch."

Create Part	
Name	PUNCH
ID	5
Color	
Ok	Apply	Cancel

2. Select **Preprocess/Elements/Copy**.

3. In the **Copy Elements** dialog box, select **Offset** in the **Type** pull-down menu. Toggle **In Original Part** off and select copy number 1. Once that is done, enter an offset thickness = blank thickness + 10 % × thickness.

4. Select **Select Element** to choose the elements. Since we only want to copy the elements that make up the U-channel of the die (not the flat surface), select the **Side View** option from the **Icon Bar.**

5. Select **Drag Window** from the **Select Elements** dialog box. Drag a window as shown below and select **OK**:
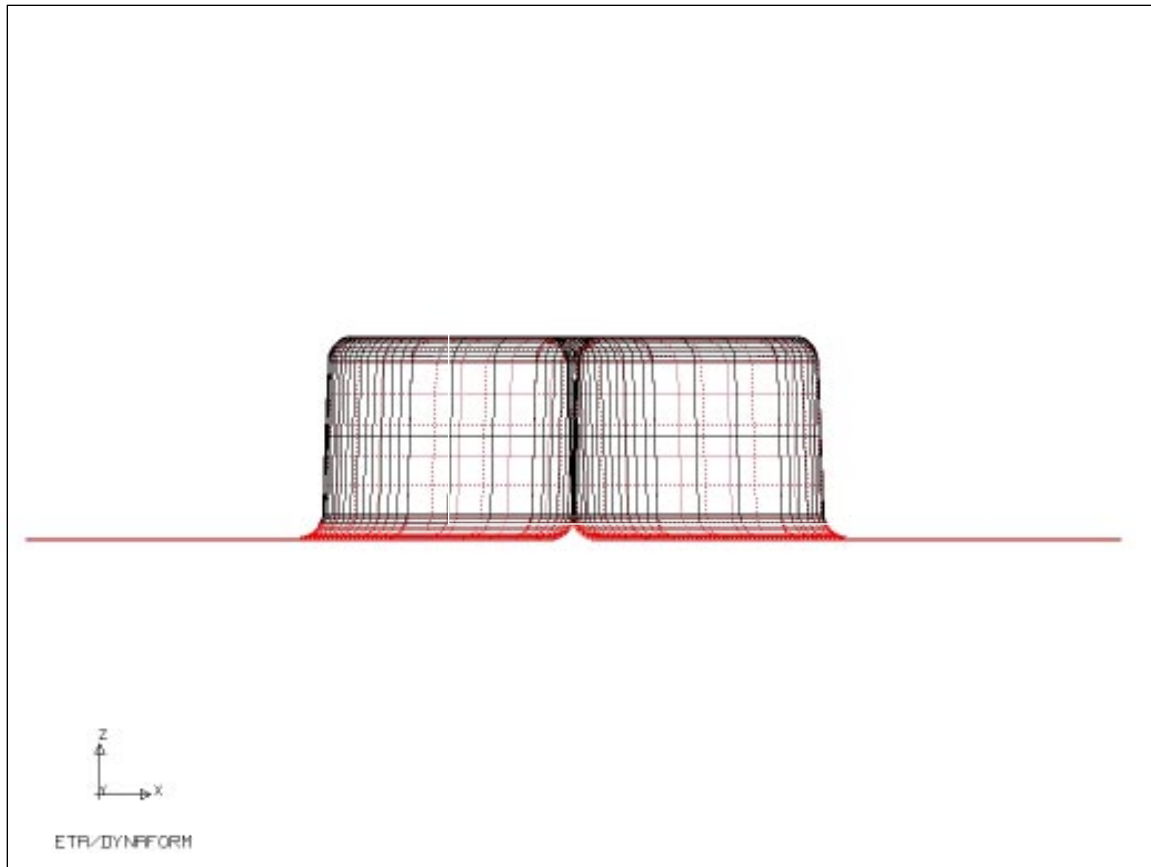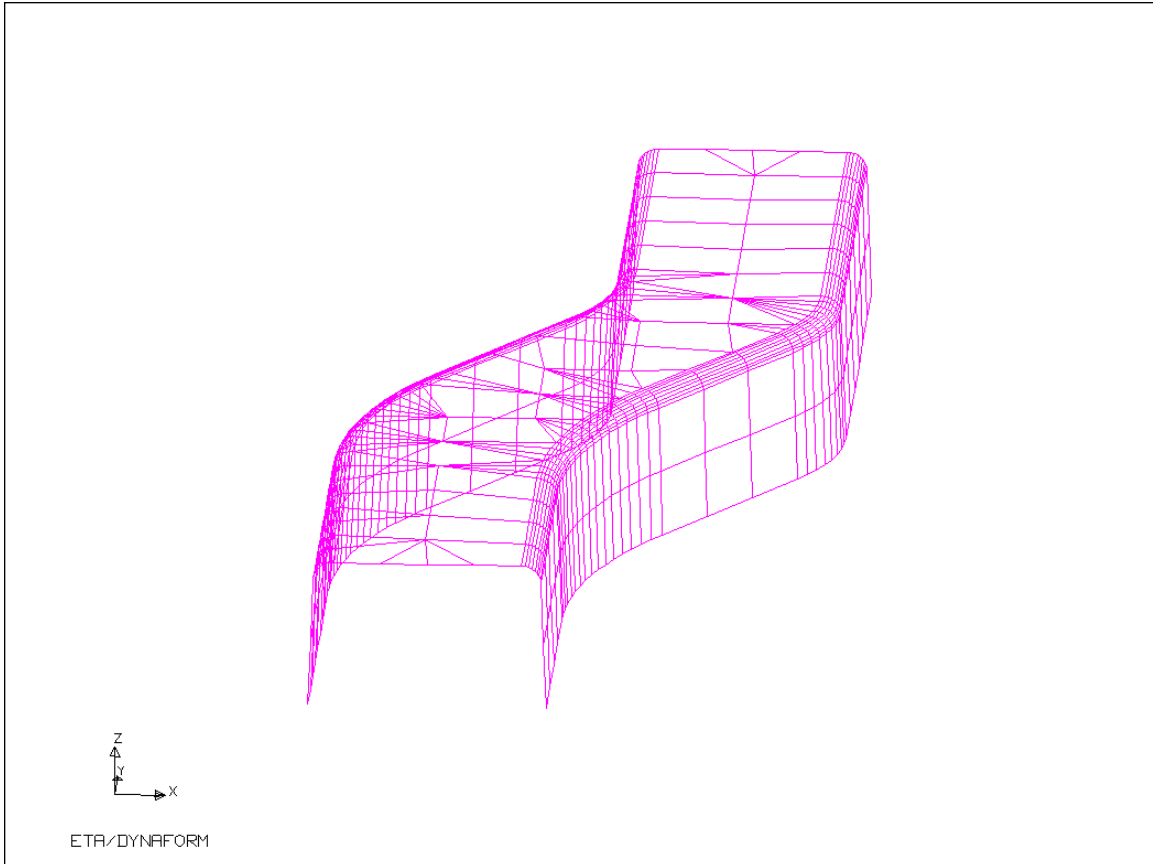
6.   Select **Apply** and click **OK**.

7.   Select **Save** to save the current changes.

    Note: By turning the Die off, you could be left with only the "Punch" model
              displayed.

8. Select **Isometric** in the **Display Options**. The punch will be displayed as the following:

ETA/DYNAFORM

Defining the Parts As Tools

Command: **Tools/Define Tools**

1. Select **Tools/Define Tool** and select **Die** in the **Tool Name** pull-down menu. Add the part called "Die.s" to be defined as the Die by using **Add.** The "Die.s" and PID will be displayed in the **Include Parts List** field.


2. Repeat the above steps for the Punch and define "BINDER.S" as the Lower Ring.

3. Select **Save** to save the changes.
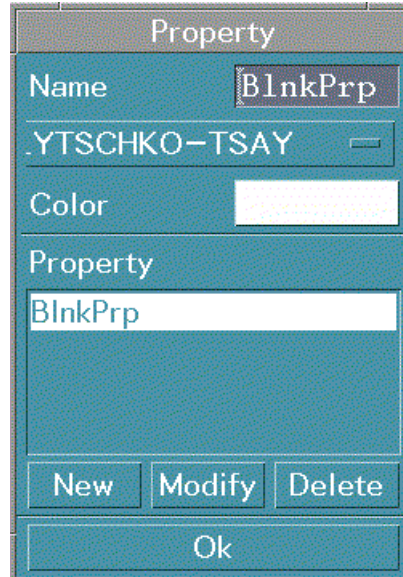
Define Blank Material/Property

Command: **Tools/Material** and **Tools/Property**

1. From the **Tools** menu, select **Material** to display the **Material** dialog box.

2. In this box, type '**Blnkmat**' for the name of the material (or any other name you like). Select **Type 36** and click **New**. The Material menu for this type will be displayed. The user can change property values by clicking on the value with the left mouse button, entering the new value and then hitting return.

3. Edit **MATERIAL PARAM (N)** to be 0.23 and click **Ok.** The name of the defined material will appear in the **Material** field.
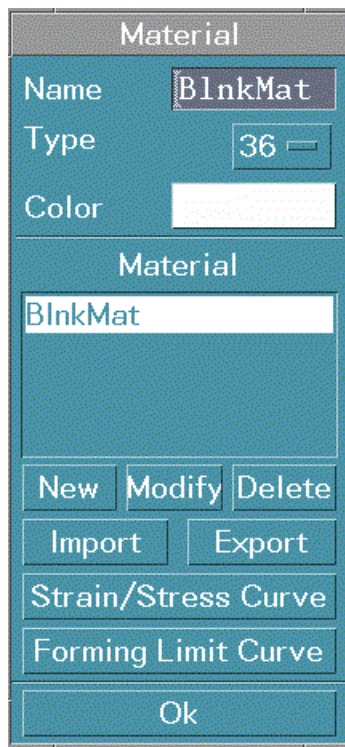
4. Next, select **Blank Property** from the **Tools** menu to display the **Property** dialog box.

5. In this box, type '**Blnkprp**' in the **Name** field.

6. Select **BELYTSCHKO-TSAY** shell element, select **New** and edit the **Uniform Thickness** field in the dialog box to be 1.0mm.


7. Click **OK** in the **Property** dialog box, the name of the defined property will appear in the **Property** field in the **Define Blank** dialog box**.**

8. Select **Tools/Define Blank** menu. Assign the material and property to "Blank".

9. Turn all parts on by **Parts/Turn on/All on**.

10. Select **Save** to save the changes.

Auto Positioning Tools

Command: **Tools/Position Tool/Auto Position**

1. eta/DYNAFORM will now adjust the spacing between Tools and Blank based on the default Blank thickness (i.e. 1.0mm).

2. Check the spacing between the tools by selecting **Auto Position/Min. Distance**.

3. In **Min. Distance between Tools** dialog box, select **BLANK** for the **First Tool** and **PUNCH** for the **Second Tool**. Select **OK** and 1.000 appears in the **Distance** field. In the prompt window, MIN. DIST. BETWEEN BLANK-PUNCH IN Z=-1.000E+01 displays.

4. Repeat the above step, this time select **PUNCH** as the **First Tool**, and **DIE** as the **Second Tool**. This distance should be 42.01mm. The distance measured is the total draw depth and will be used in the next section to generate the punch velocity curve.
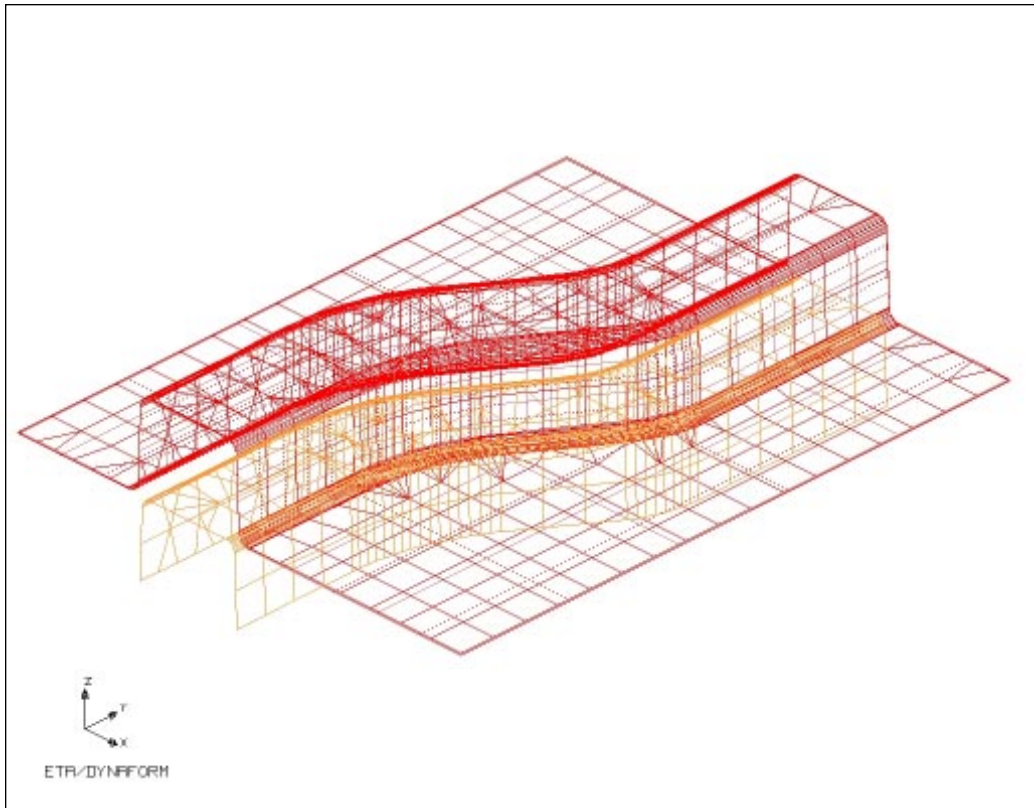
Define Contact Parameters and Punch Velocity Curve
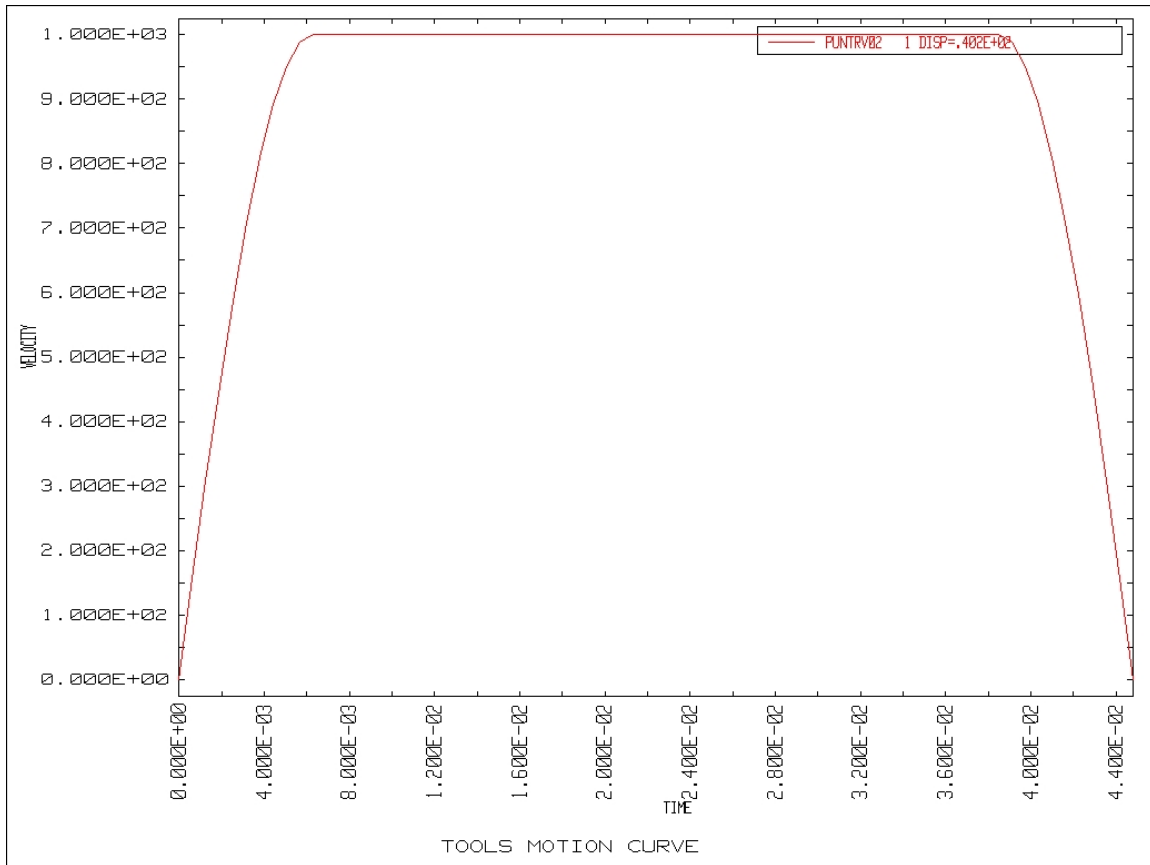
Command: **Tools/Define Tools /Define Contact**.

1. Select **Die** in the **Tool Name** pull-down menu for defining the Die contact.

2. Select contact type **Forming-One-Way-S-S**.

3. Edit the **Static Friction** value to be 0.11 in the parameter dialog box and select **OK**.
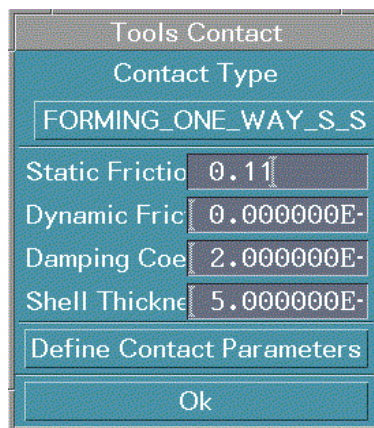
4. Repeat the steps for the Punch and the Lower Ring by selecting their respective name in the **Tool Name** list.

5. Select **Punch** in the **Tool Name** pull-down menu, then select **Define Motion/Auto Create Curve** and click **Done**.
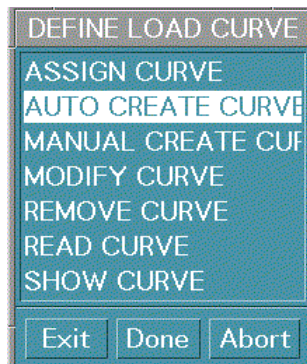
6. In the **Velocity/Time Curve** dialog box, type '1000'mm/s for velocity, and '41.01'mm for stroke distance. The stroke distance = the distance measured previously between the Punch and the Die '42.01'mm -the Blank thickness 1.0mm=41.01mm.

7. Select **Full Screen** to display the punch velocity curve as shown below.
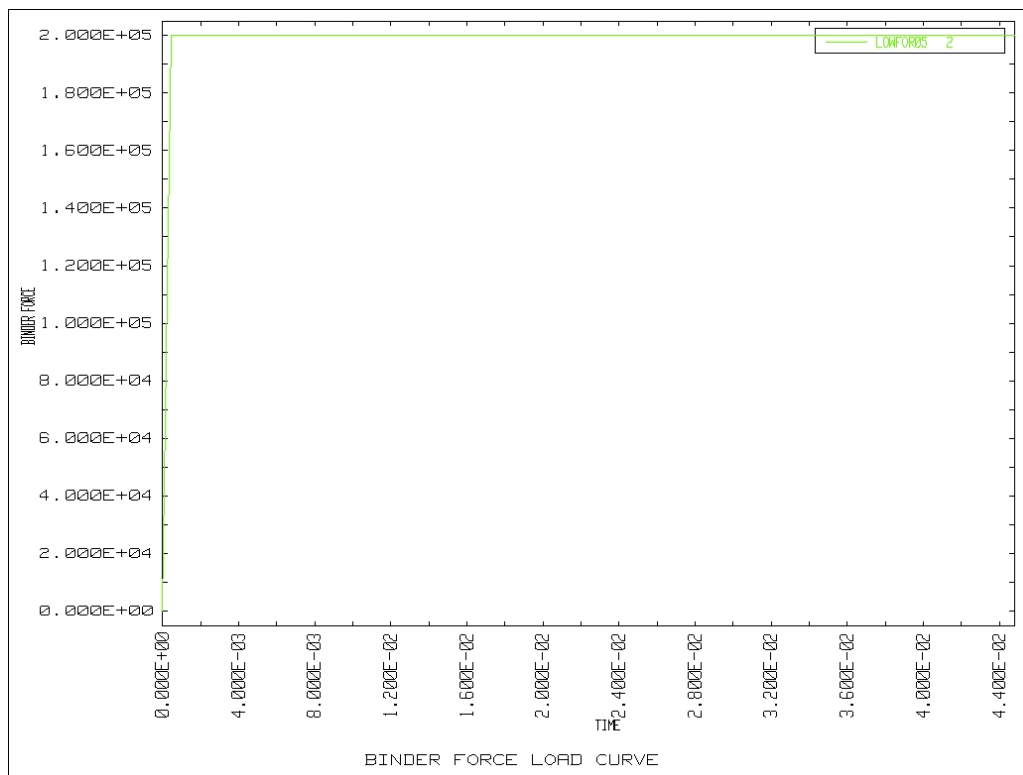

8. Select **Done** and then **Exit**.

Defining Lower Ring (Binder) Force Curve

1. From **Tools/Define Tools**, select **Lower Ring** in the **Tool Name** pull-down menu.

2. Select **Define Force/Auto Create Curve** and click **Done**.

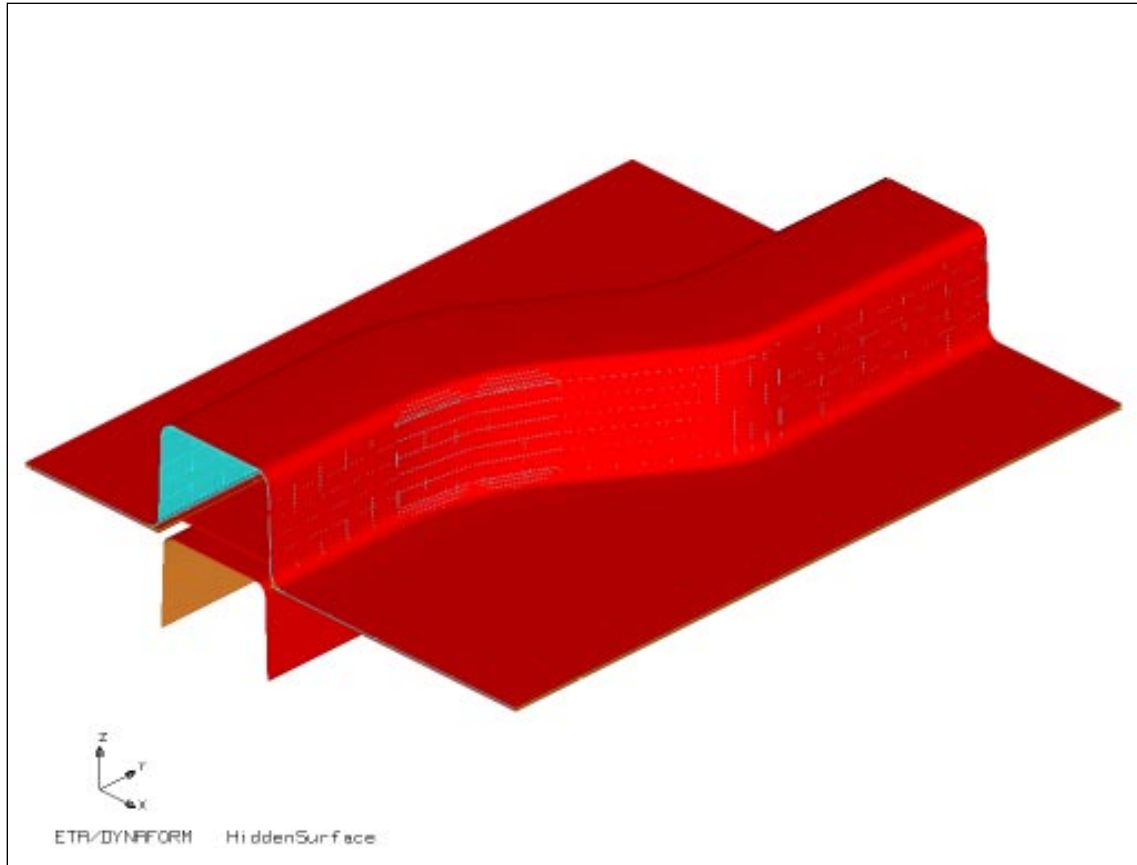3. Type '200000' N in the force field and click **Ok**.
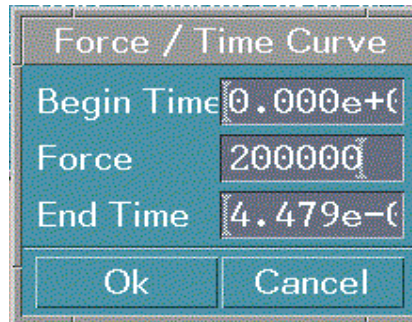
4. Select **Full Screen** to display the Binder Force Load Curve as the following.

Preview Tool Animation

Command: **Tools/Animate**

Before running the analysis, the user should preview the tools' animation. Select **Tools**/**Animate** to see the results.

Run Analysis

1. Open the dialog box **Analysis** from the **Analysis** menu.

2. Select **DYNA940 Input File** for **Analysis Type.**

   Note: The user has the option of two analysis types: **Dyna950 Input File** and **Dyna940 Input File**. There is a slight difference between the **Dyna950 Input File** and the **Dyna940 Input File** dialog boxes. The user can compare their contents with the following two figures. If the user selects **Dyna940 Input File**, the user will toggle the button **Springback** to get a NIKIN file after running. If the user selects **Dyna950 Input File**, the user will toggle the **Seamless** button in the **Analysis** menu. The **Seamless** function can directly handle trimming and springback when running. For more detailed information, please refer to the User's Manual.

3. Check the **Control Parameter** menu to make sure all of the parameter values are acceptable.

4. If **Adaptive Mesh** is needed, the user can open the associated dialogue box and check the values of the parameters. In this case, the user need not use it.

5. Type '**dftrain**' in the **File** field and '**DYNAFORM training exercise**' in the **Title** field.

6. The input files will then be written and the user can submit them to LS-DYNA.

Note: Two files are written out, dftrain.dyn and dftrain.mod. Dftrain.mod contains all of the model data which is referenced by an 'include' statement in the dftrain.dyn file.

Note: Once the job is submitted, type 'ctrl-z' in the window where the job is running. Then type 'bg' once the Unix prompt appears. This will put both Dyna and Dynaform in the background. Then, to update the job runtime, simply type 'kill -2 ps', where ps=dyna process #. If a Sense Switch other than SW2 is desired, create a text file called 'switch' with that particular switch listed (i.e. sw1, sw3 etc.). This switch file should be located in the directory where you enter the 'kill -2' command. Now, instead of the default sw2 switch being used, the command saved in the switch file will be used.

Note: The analysis can also be submitted from the Unix command line by typing 'dyna i=input filename.' Dyna is the alias to run LS/DYNA and the input filename would be dftrain.dyn in this case. If more memory is required, the command 'dyna i=input filename memory=# of words short' can be used to submit the job and reallocate the required memory, where '# of words short' is the additional memory required in bytes.

Once the job has finished running, we will post-process the results in eta/DYNAFORM and trim the Blank before running the Spring Back analysis. We will then post-process those results in eta/DYNAFORM.

Post Processing Forming Results

eta/DYNAFORM can read and process all the available data in the D3PLOT file. In addition to the undeformed model data, the D3PLOT file also contains all requested result data (stress, strain, time history data, deformation, etc.) from LS-DYNA. eta/DYNAFORM will automatically create a new database called 'd3plot.df' into which we will read in the D3PLOT file directly.

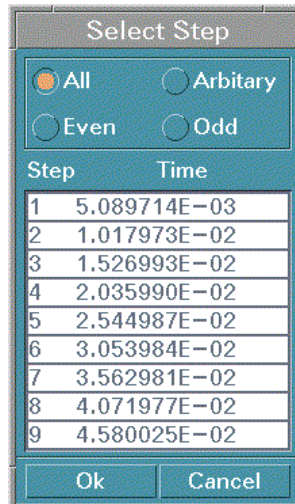## Post Processing – Reading a D3plot File

1. Select **Open** in the **File** menu and the **Open File** dialog box is displayed.

2. Toggle **Postprocess File** on and choose **d3plot**.

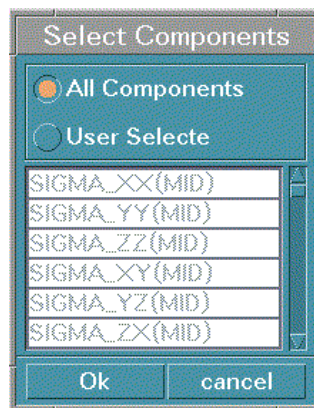3. Select **d3plot** in the **File** field and click **OK**.
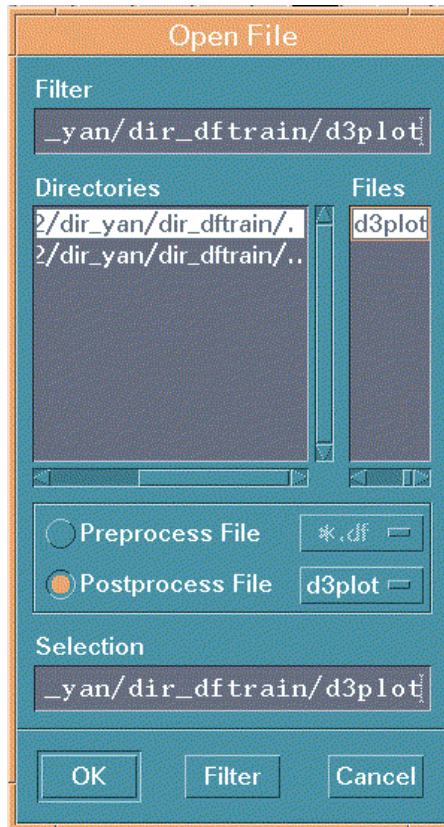
4. Select **LS940** from the **SELECT DYNA3D VERSION** dialog box.

5. In the dialog box, **Select Step**, select the step you need. For this exercise, toggle the button **All**.

6. Select **All Components**. All of the D3PLOT files will be read into eta/DYNAFORM.

7. Keep only the Blank on by using **Parts/Turn on**.

8. Select **Isometric View** from the **Icon Bar**.

Post Processing-Animating a Contour

1. Select **Postprocess/Animate Contour** and the **Animate Contour** dialog box is displayed.

2. Select **Others** from the **Components** pull-down menu and then select the **Thickness** component.
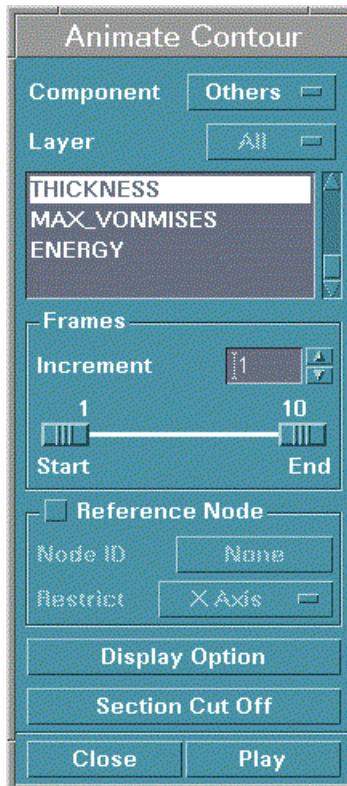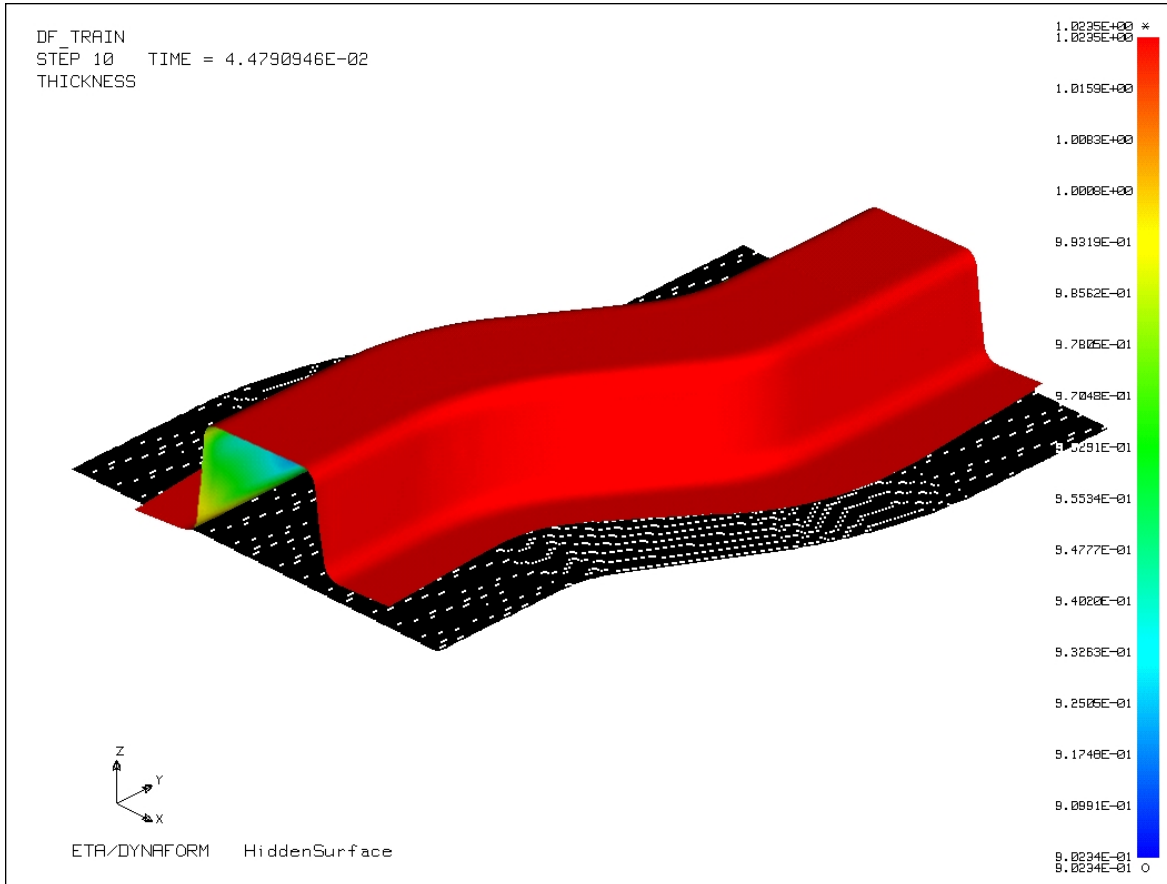
3. Select **Display Option,** set the **Contour Color Type** to **Continue** in the **Display Option** pull-down menu and select **OK**.
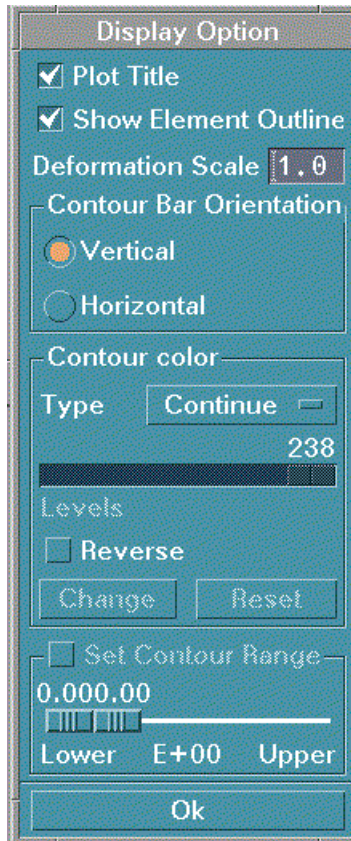
4. Toggle **Shade** at the right hand corner of the screen in the Display Options.

5. Select **Play**. eta/DYNAFORM will construct and animate the thickness contour plots for all 10 times steps. The contour plot for the 10th time step is shown below.


6. The user can see the individual frame by toggling **Individual Frame** on and to discontinue the animation, select **Stop** from the **Animate** dialog box.

7. Select **Close** in the **Animate Contour** dialog box.

Post Processing-Plotting a Contour/Listing a Contour Value

Command: **Postprocess/Plot Contour**

1. Select **Postprocess/Plot Contour**.

2. Select Step #10 with the **Current Frame** slider, select **Others** from the **Components** pull-down menu and select the **Thickness** component.

3. Click **Plot**. A contour plot is displayed similar to the one on the previous page.
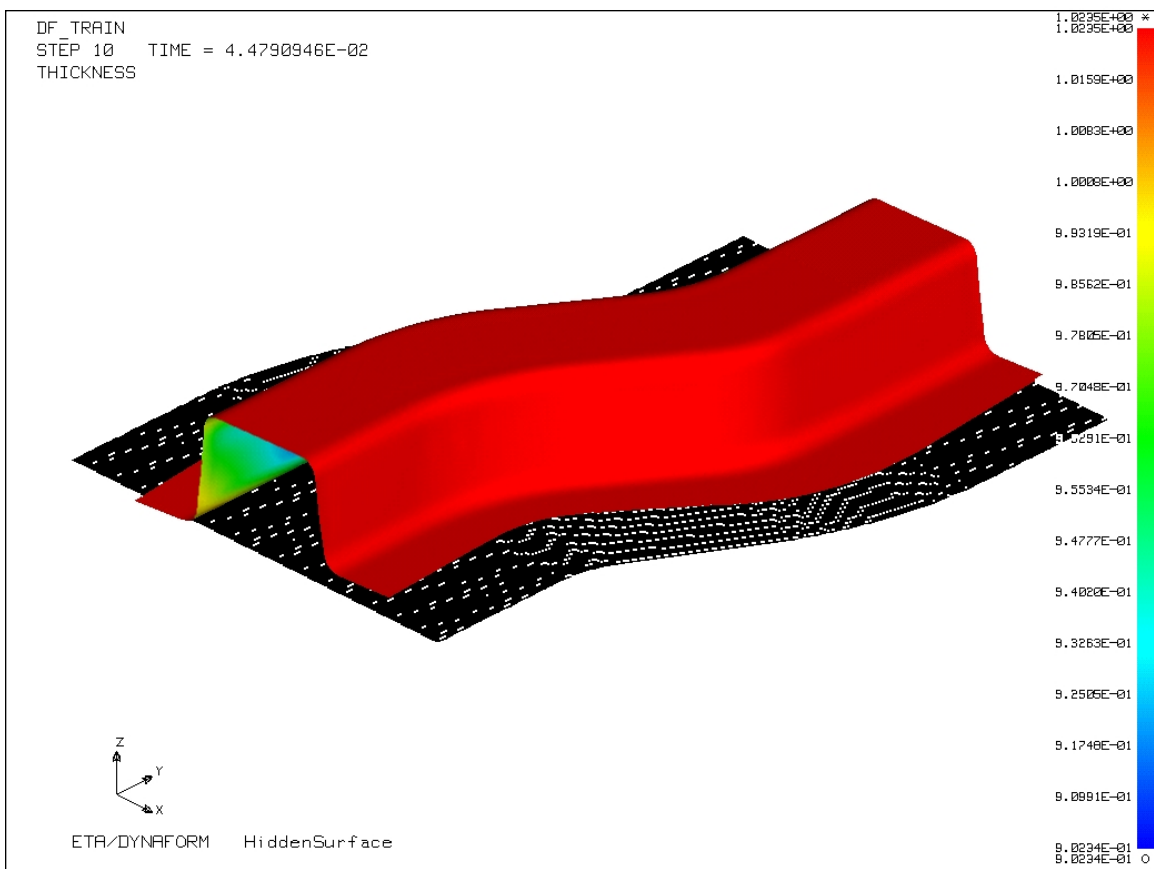

4. Select **List Contour Value**.

5. The default setting is **Select By Node**. Pick a node on the model by clicking the left mouse button.

6. A table appears in the lower left region of the display area indicating the thickness value for that node.

INDEX	NODE ID	THICKNESS
1	6162	0.9400

7. Select **Profile**. This option displays the contour value for all the nodes in the model and lists them in a table. The table size is limited to 10 values. Scan through the table by using the **Browse Value** commands to go to the previous and next table listings respectively.
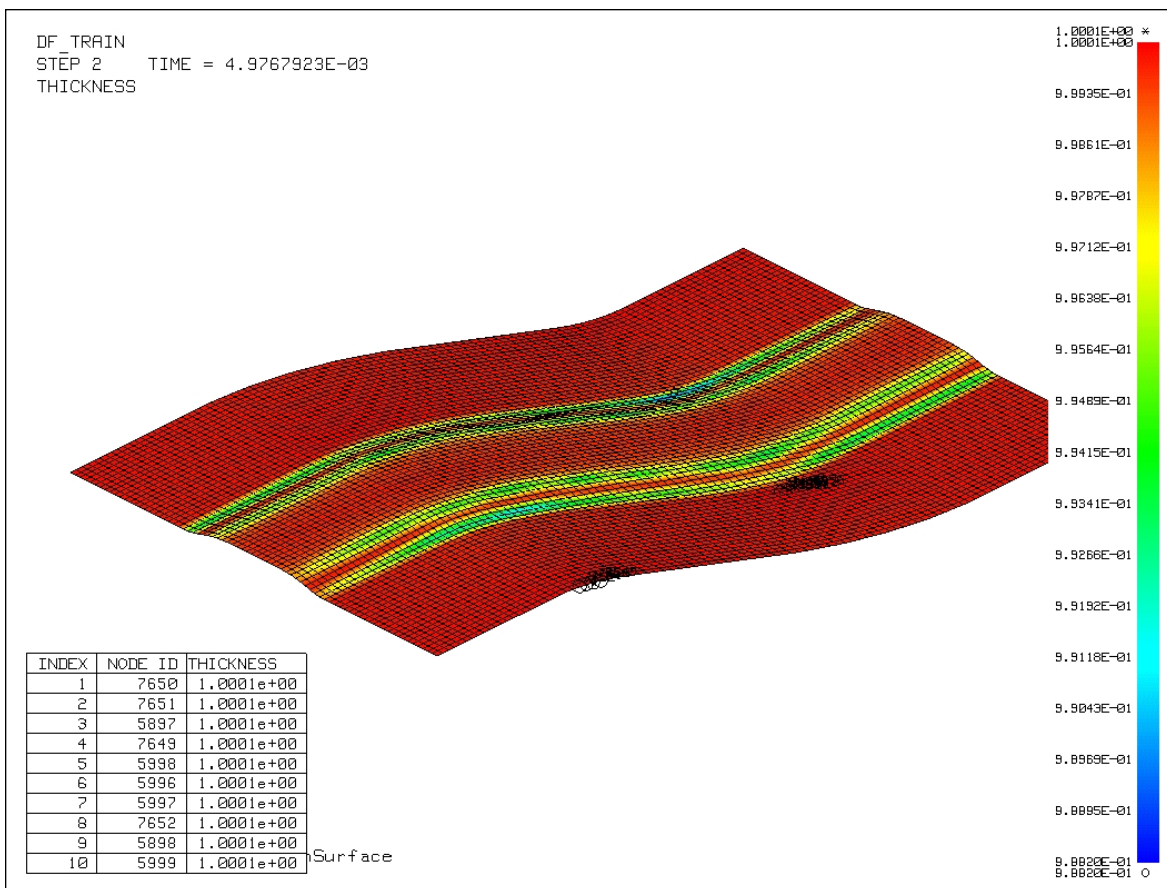
Post Processing - Plotting a Contour with a Forming Limit Diagram

This feature displays a contour plot based on a Forming Limit Diagram that is read in, or defined by the user. The Forming Limit Diagram is based on material properties N, R, and T (where N = Resistance to Necking, R = Anisotropic Hardening Parameter, and T = Thickness).

1. Select **Postprocess/Plot FLD**.

2. Select **Engineering** in **Strain Type**.

3. Select **Defined by n, r, t** in FLD Curve and set the r value to e 1.04. Accept the default values n=0.23 and t=1.0.

4. Select the **Middle** surface in the **Layer** pull-down menu.

5. Choose the 10th time step in the **Current Frame**.

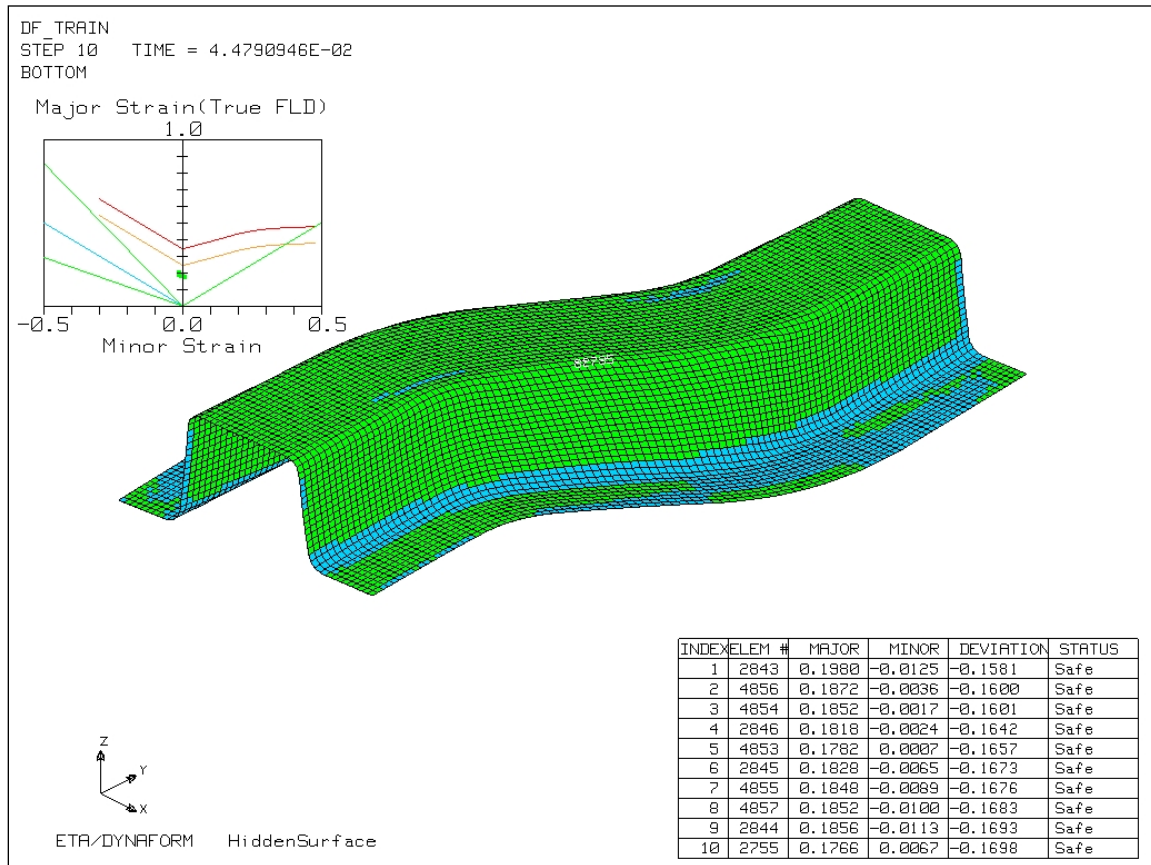6. Select **Plot**.

7. The FLD will be displayed in the left upper corner.
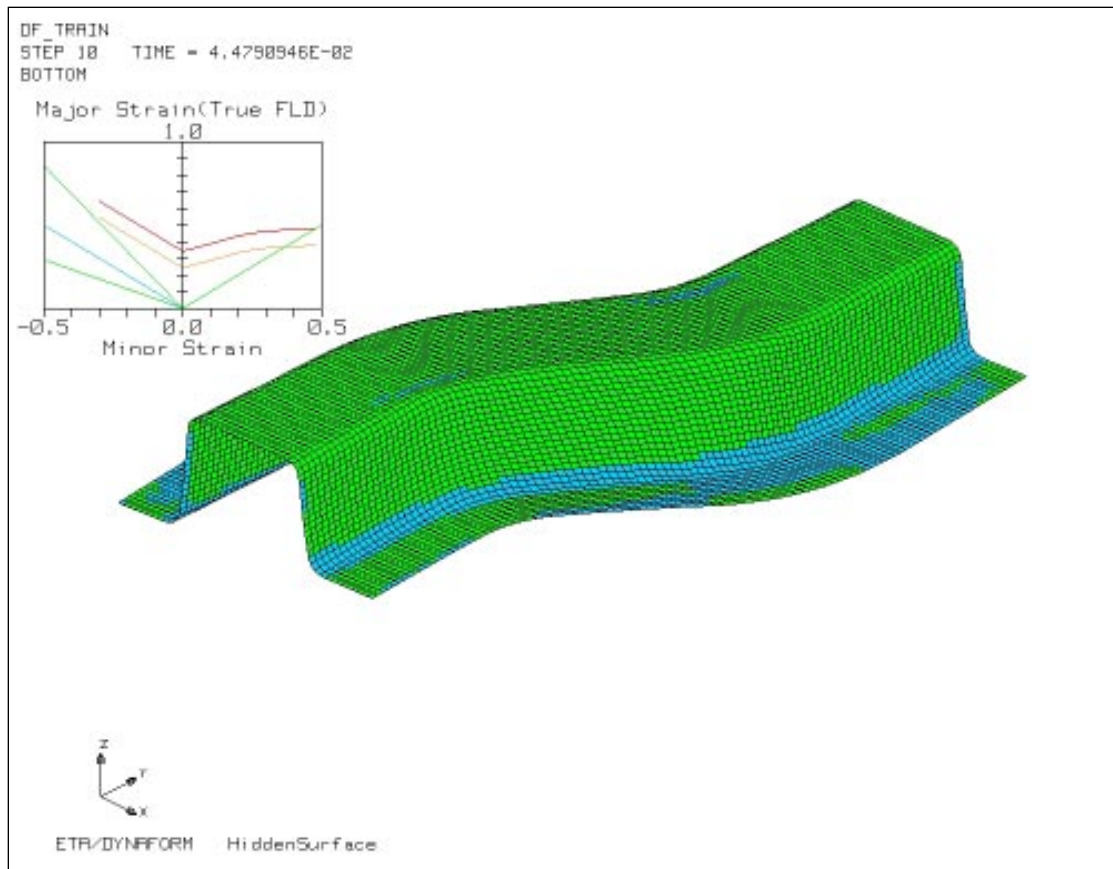
8. Select **List FLD Value** in the Plot FLD dialog box and select **ALL**. This will plot a contour based on the FLD curve defined and will show all the elements locations on the curve. This will also tabulate the 10 elements with the highest strain values along with their formability status (i.e. Buckling/ Marginal/ Critical/ Fail), as shown below.


```
DF_TRAIN
STEP 10    TIME = 4.4790946E-02
BOTTOM

 Major Strain(True FLD)
             1.0




-0.5       0.0       0.5
    Minor Strain




    z
      Y
        x

ETA/DYNAFORM    HiddenSurface
```

INDEX	ELEM #	MAJOR	MINOR	DEVIATION	STATUS
1	2843	0.1980	-0.0125	-0.1581	Safe
2	4856	0.1872	-0.0036	-0.1600	Safe
3	4854	0.1852	-0.0017	-0.1601	Safe
4	2846	0.1818	-0.0024	-0.1642	Safe
5	4853	0.1782	0.0007	-0.1657	Safe
6	2845	0.1828	-0.0065	-0.1673	Safe
7	4855	0.1848	-0.0089	-0.1676	Safe
8	4857	0.1852	-0.0100	-0.1683	Safe
9	2844	0.1856	-0.0113	-0.1693	Safe
10	2755	0.1766	0.0067	-0.1698	Safe

9. Select **OK** in the **List FLD Values** dialog box.

10. Select **Close** in **Plot FLD** dialog box.

11. Exit **Post Process**.

Trimming the Blank

eta/DYNAFORM has a trimming function that allows the user to trim the blank before running a Spring Back analysis. For this exercise, we will read the NIKIN file into eta/DYNAFORM, view the formed blank shape, trim the blank based on a given trim line, submit the job to LS-NIKE, and then post-process the results.

1. Create a new database called 'dftrain3.df'. Select **Draw Type** - **Springback**.

2. *Command*: **File/Import**

   Select **File/Import** and choose **Nike3D Input** in the **Forma**t pull-down menu. Select the file **nikin.nik**. Repeat the above step to import trimline.lin file in the **Line Data** format into eta/DYNAFORM. The user will get the following figure.
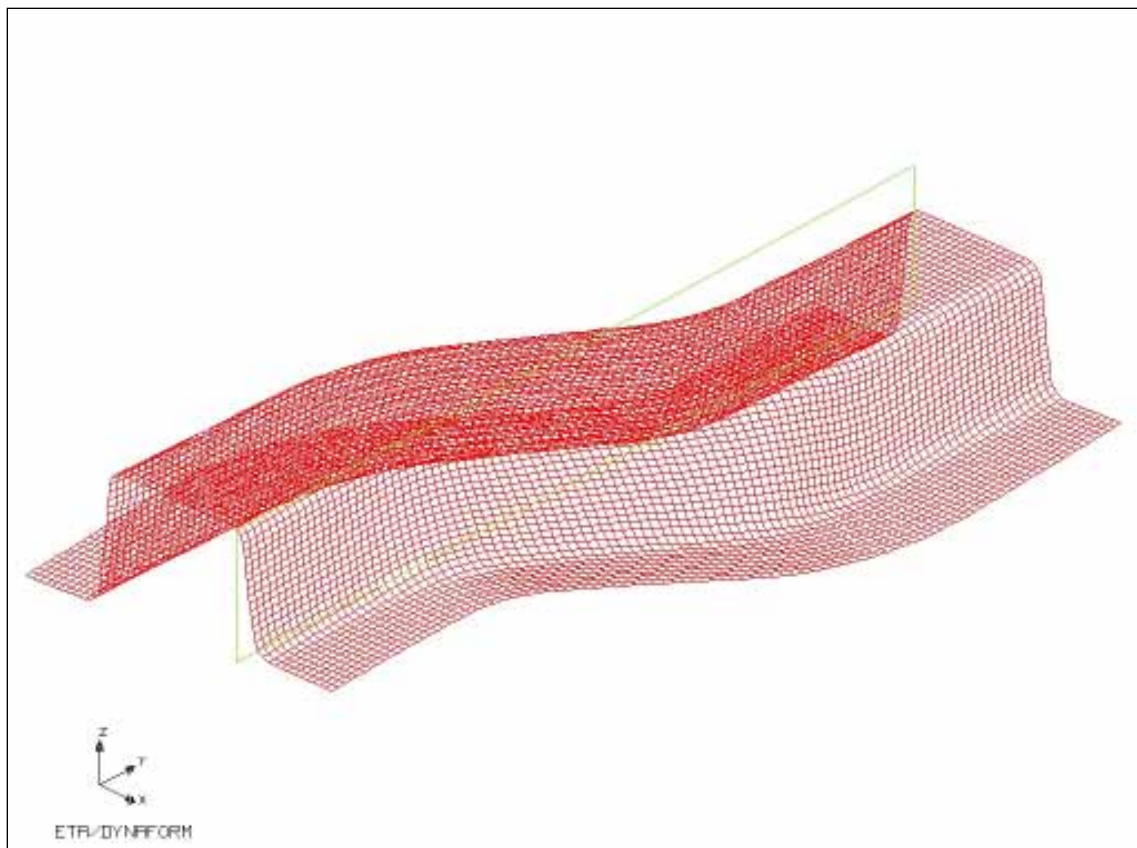
3. *Command*: **Tools/Blank Miscellanies/Trim**

The dialog box will appear on the screen. Choose the trim line on the screen and select **OK**.
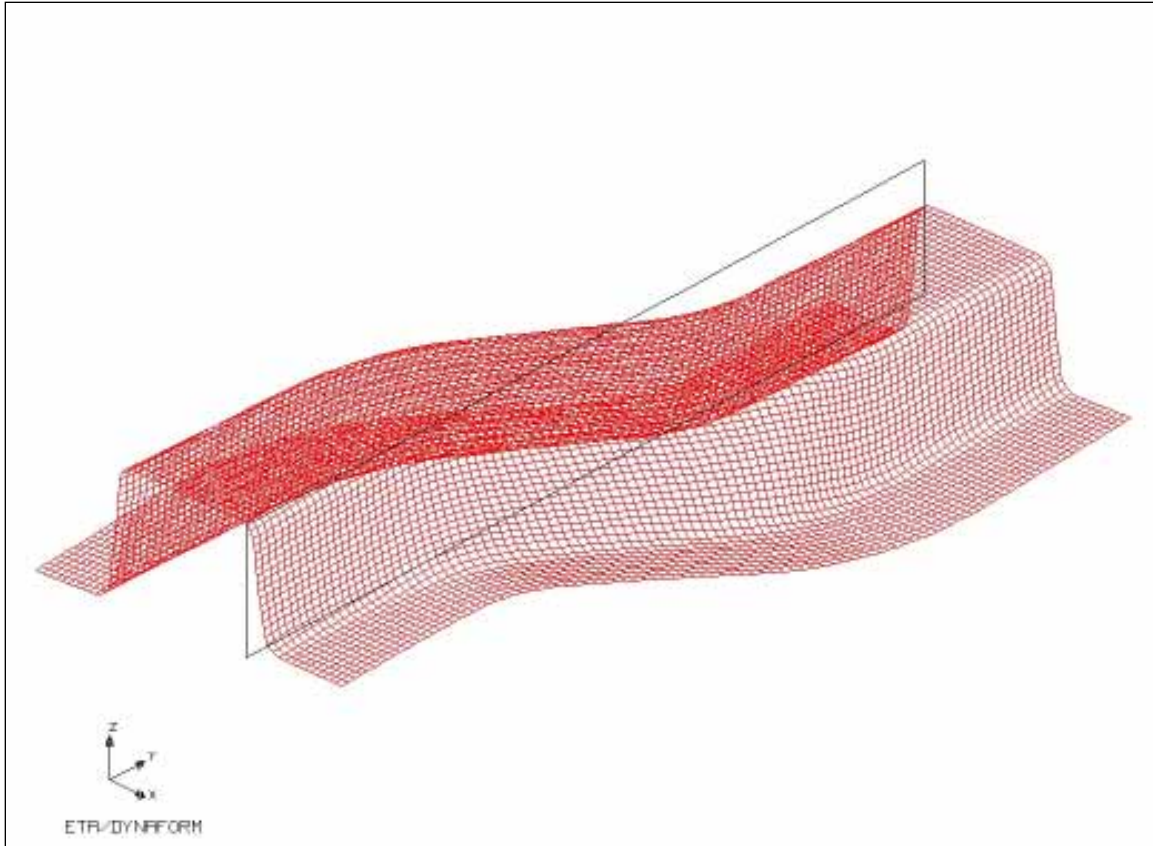
4. Another dialog box appears on the screen for the user to select the direction of the trimming and eta/DYNAFORM prompts the user to select a node for the origin. Select one node and toggle **Define By 3 Location** off in the **LCS** dialog box to enable the **U**, **V**, and **W Axis** buttons.
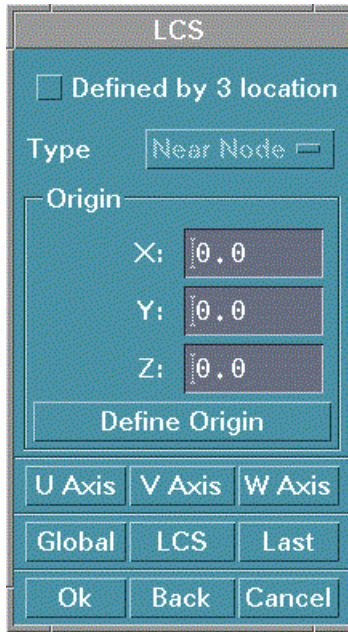
5. Select the **W Axis** from the **LCS** dialog box as the trimming direction and press **OK**. The **W Axis** dialog box appears:
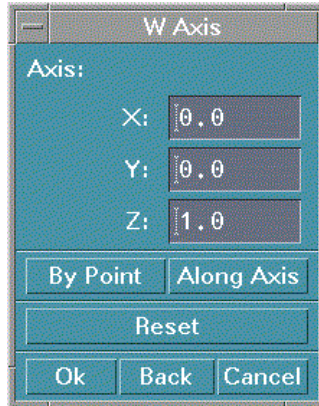
6. Select **Along Axis** from the **W Axis** dialog box and the following dialog box will appear. Select **Along X** and **Done**.

7. The **W Axis** dialog appears. Select **OK**. Then, the **LCS** dialog box appears. Press **OK** again. The **Tolerance** dialog will then be displayed. Accept the default value in the **Tolerance** dialog box.

8. A text window appears on the screen. eta/DYNAFORM prompts:
    > EXCLUDE ELEMENTS FROM TRIM LINE?
   Select **Yes** and see the result of the trimming.
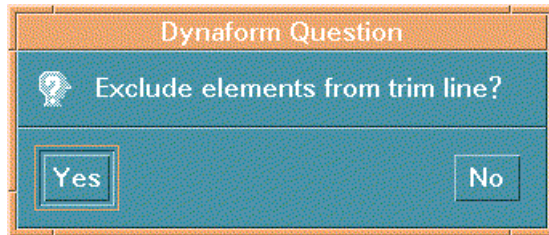
9. Open **Part/Turn On** and turn off TRIMLINE and TRIMOUT. This enables the user
   to see the trimming result in part MAT00002.
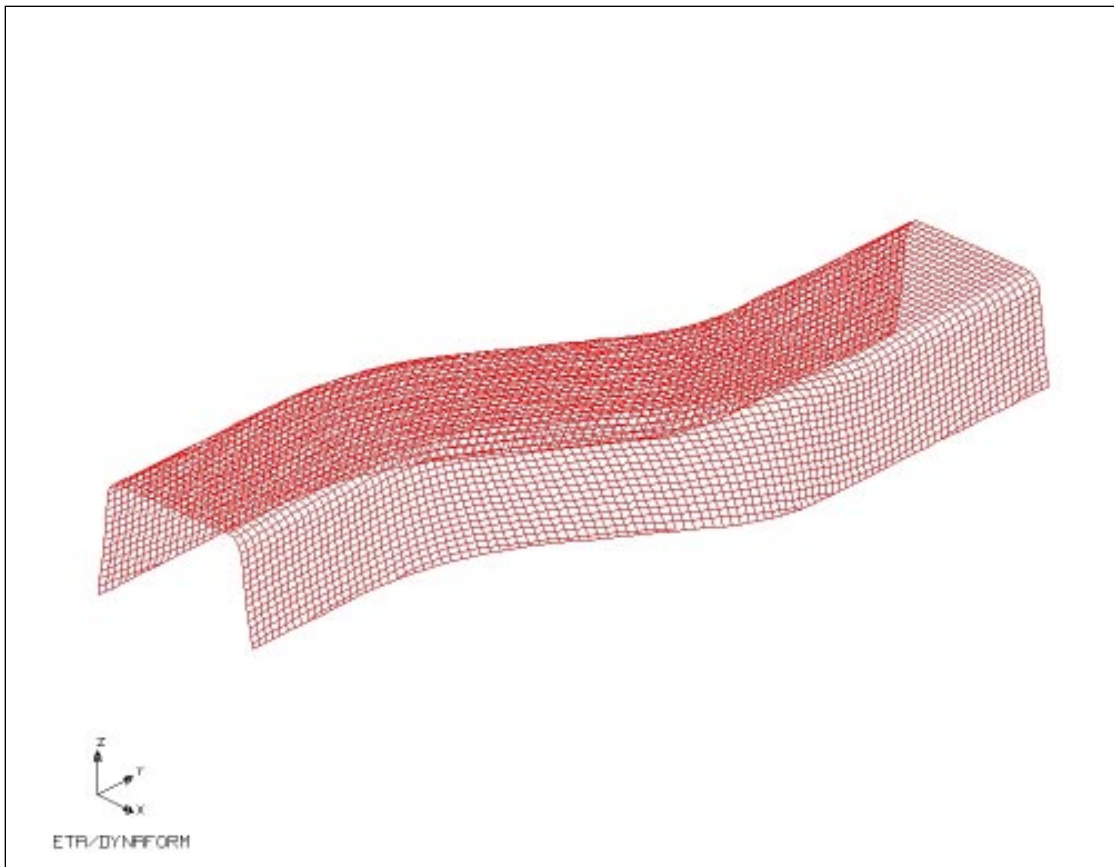
Since we will be running a linear, static analysis, some locations must be constrained
on the Trimmed Blank model to eliminate rigid body motion. We will do this by using
Single Point Constraint (SPC) to constrain the minimum 3 points. The constraint
sequence, shown in the following figure, is based on the global coordinate system.
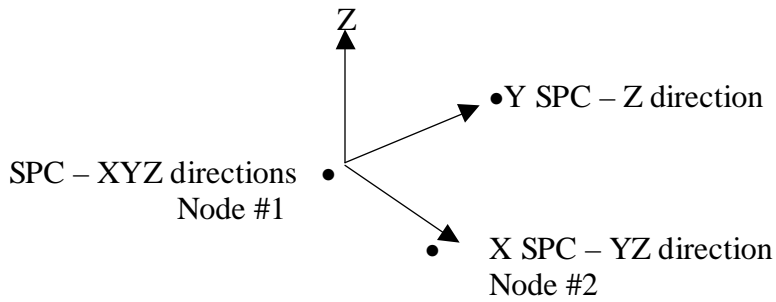
10. The user should select nodes on the surface of the blank to constrain the rigid motion. Change the view to **Top View**. Select **Preprocess/Boundary Conditions** and a dialog box is displayed on the screen. Choose **SPC OPTION,** select **Done** and the **SPC Set** dialog box displays.
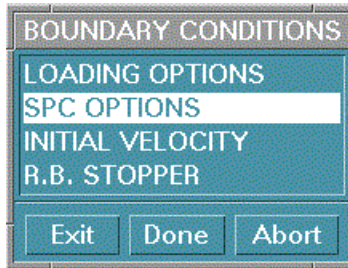
11. The following is displayed:
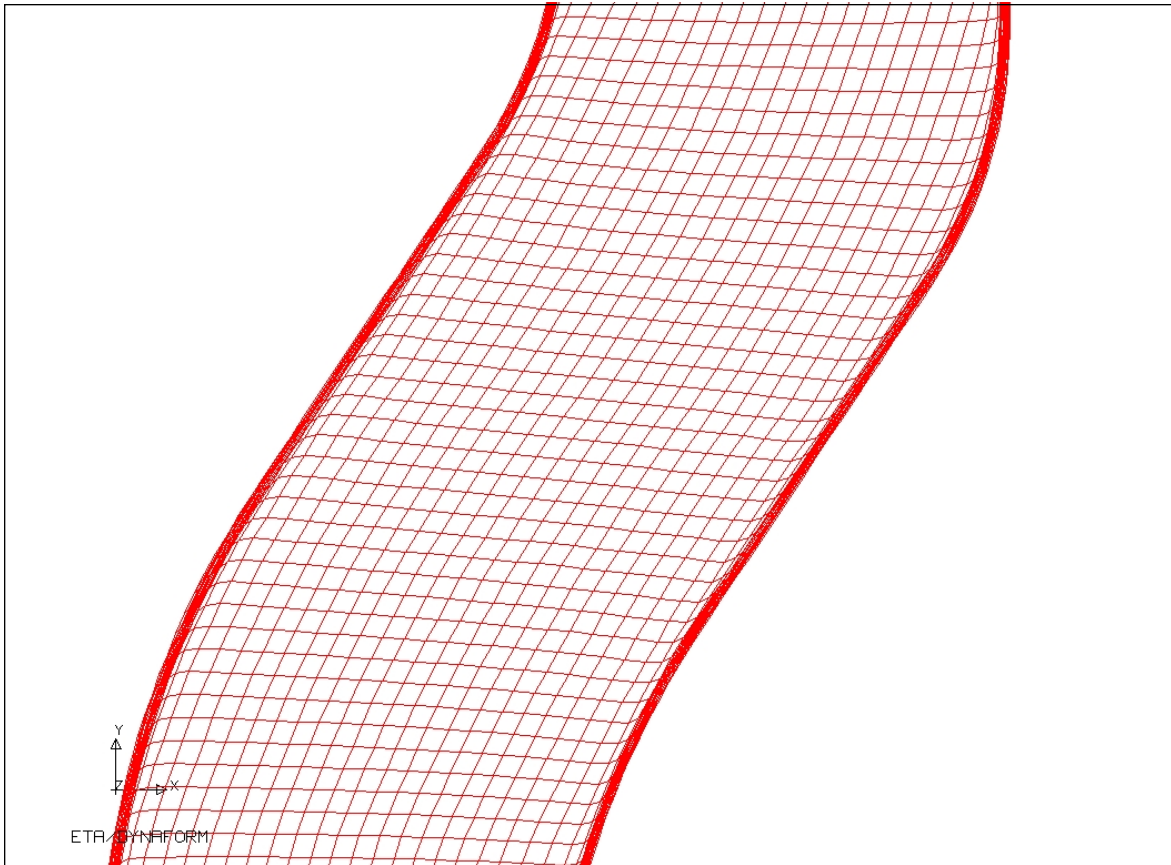
Select **Create** to create three points on the center of the blank surface in terms of the figure. These three points can be removed from the rigid body motion when the user needs to perform the springback portion of the analysis. Select three points as describe in the SPC figure on the previous page. The position of the three points should be in the center of the blank surface.
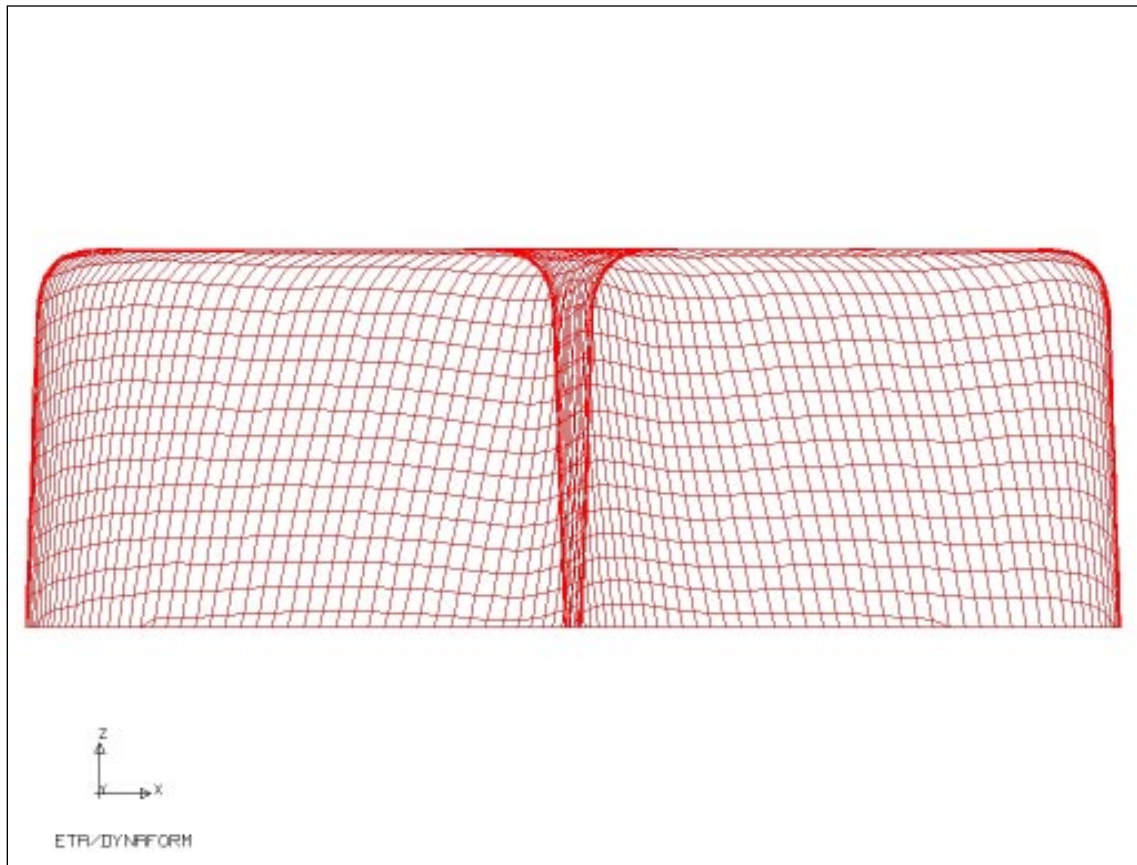
12. Select **OK** in the **SPC Set** dialog box and the **Boundary Conditions** dialog box displays.
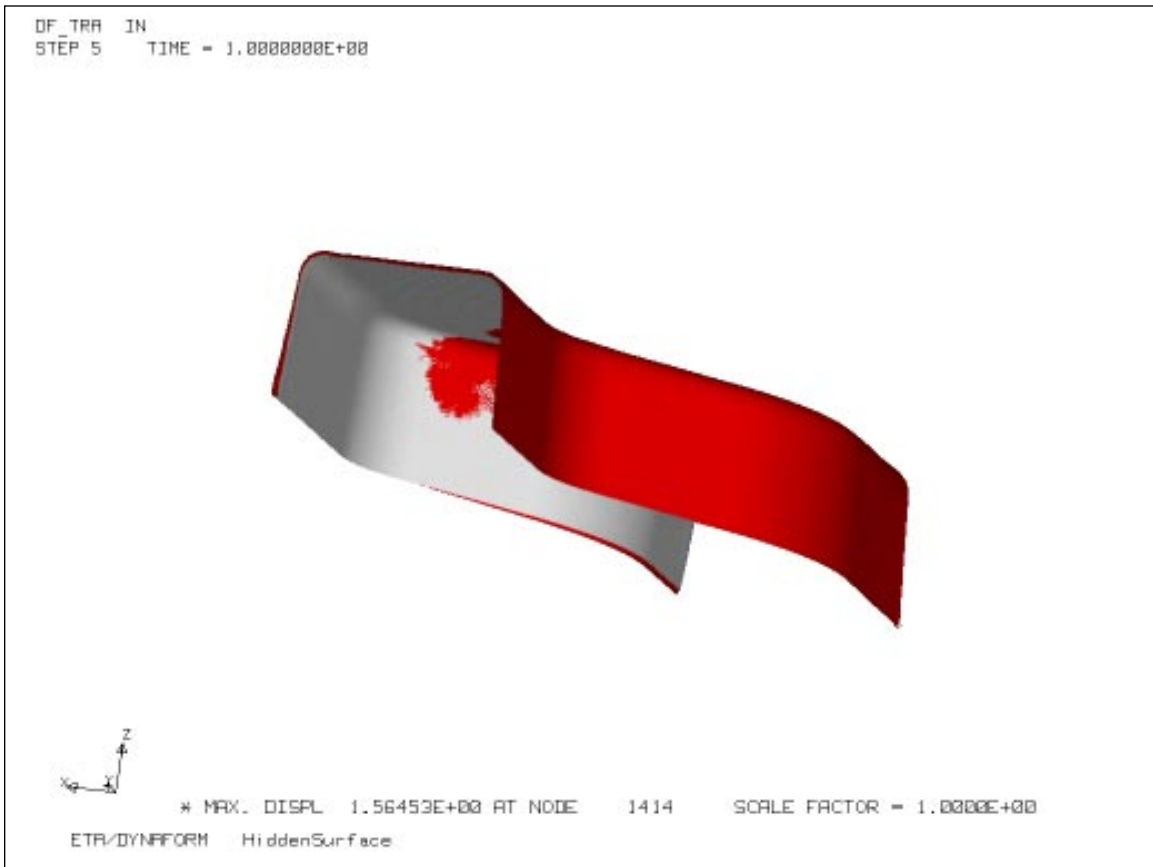
13. Select **Exit**.

Post Processing Springback Results

eta/DYNAFORM can also read and process all the available data in the N3PLOT file. The undeformed geometry is contained in this file along with the results.

1. Select **File**/**Open**.

2. Toggle **Postprocess** File on and then choose **n3plot.**

3. Select **n3plot** in the **File** field and click **OK**.

4. Repeat step 4 – 8 of **Post Process - Reading d3plot File**.

5. Select **Postprocess**/**Animate Deformation** to see the results of the springback analysis.

6. Select **Postprocess** / **Plot Deformation.**


```
DF_TRA  IN
STEP 5    TIME = 1.0000000E+00
```

```
      z
      |
x     |
  \___|___ y


        * MAX. DISPL  1.56453E+00 AT NODE    1414    SCALE FACTOR = 1.0000E+00

   ETA/DYNAFORM   HiddenSurface
```

Conclusion

This concludes the training guide's basic overview of eta/DYNAFORM. The user should now have the basic skills necessary to create models, prepare models for analysis and display result data. Please reference the User's Manual for more detailed explanations of each function.