

# eta/DYNAFORM User's Manual

Version 5.5



Engineering Technology Associates, Inc.  
1133 E. Maple Road, Suite 200  
Troy, MI 48083-2896

Phone: (248) 729-3010  
Support: (800) 382-3362  
Fax: (248) 729-3020

Engineering Technology Associates, Inc., ETA, the ETA logo, and eta/DYNAFORM are the registered trademarks of Engineering Technology Associates, Inc. All other trademarks or names are the property of the respective owners.

# TABLE OF CONTENTS

---

---

<b>INTRODUCTION</b> .....	<b>1</b>
<b>STRUCTURAL OVERVIEW</b> .....	<b>5</b>
1.1 GENERAL .....	5
1.2 MENU BAR .....	5
1.3 ICON BAR .....	6
1.4 DISPLAY WINDOW .....	9
1.5 DISPLAY OPTIONS .....	11
1.6 MOUSE FUNCTIONS .....	13
1.7 SPECIFICATIONS .....	14
1.8 GEOMETRY DATA .....	14
1.9 RECOMMENDED NAMING CONVENTION .....	14
1.10 DIALOG BOXES .....	15
1.11 PROPERTY TABLES .....	16
1.12 CONFIGURATION FILE .....	17
<b>GETTING STARTED</b> .....	<b>19</b>
2.1 GETTING STARTED .....	19
2.2 STARTING WITH CAD DATA .....	19
2.3 GETTING STARTED WITH MODEL DATA (NASTRAN, LS-DYNA, ETC.) .....	20
2.4 GETTING STARTED FROM SCRATCH .....	20
2.5 LOCAL COORDINATE SYSTEM .....	22
<b>FILE MANAGER</b> .....	<b>25</b>
3.1 NEW .....	26
3.2 OPEN (CTRL+O) .....	26
3.3 RESTART .....	27
3.4 SAVE .....	27
3.5 SAVE AS .....	28
3.6 IMPORT (F3) .....	28
3.7 EXPORT (SHIFT+F3) .....	29
3.8 SUBMIT DYNA FROM INPUT DECK .....	30
3.9 RUN DYNA RESTART .....	32
3.10 PRINT SETUP .....	33
3.11 PRINT .....	36
3.12 EXIT (CTRL+X) .....	36
<b>PART CONTROL</b> .....	<b>37</b>
4.1 CREATE PART (CTRL+P) .....	38

4.2	EDIT PART .....	39
4.3	DELETE PART.....	40
4.4	ADD... TO PART.....	41
4.5	TURN ON .....	47
4.6	CURRENT PART .....	48
4.7	SEPARATE PART .....	49
4.8	TRANSPARENT.....	49
4.9	SUMMARY .....	49
<b>PREPROCESS .....</b>		<b>51</b>
5.1	LINE/POINT (CTRL+L).....	51
5.2	SURFACE (CTRL+S) .....	71
5.3	ELEMENT (CTRL+E).....	89
5.4	NODE (CTRL+N) .....	112
5.5	MODEL CHECK & REPAIR (CTRL + R).....	122
5.6	BOUNDARY CONDITIONS (CTRL+U).....	132
5.7	NODE/ELEMENT SET (CTRL+V) .....	142
<b>DFE (DIE FACE ENGINEERING) .....</b>		<b>144</b>
6.1	PREPARATION .....	145
6.2	BINDER.....	180
6.3	ADDENDUM DESIGN.....	211
6.4	RE-ENGINEERING.....	248
6.5	MODIFICATION .....	256
6.6	DIE DESIGN CHECK.....	272
<b>BSE (BLANK SIZE ENGINEERING) .....</b>		<b>274</b>
7.1	PREPARATION .....	275
7.2	MSTEP.....	277
7.3	DEVELOPMENT.....	279
<b>SETUP.....</b>		<b>302</b>
8.1	QUICK SETUP (QS) .....	302
8.2	AUTO SETUP .....	317
<b>TOOL DEFINITION .....</b>		<b>399</b>
9.1	ANALYSIS SETUP.....	400
9.2	DEFINE TOOLS (CTRL+T) .....	402
9.3	POSITION TOOLS .....	411
9.4	DRAWBEAD (CTRL+D).....	413
9.5	BLANK GENERATOR .....	433
9.6	GUIDE & PIN .....	436
9.7	DEFINE BLANK (CTRL+B).....	440



9.8	BLANK OPERATION.....	442
9.9	MATERIAL.....	447
9.10	PROPERTY.....	455
9.11	ANIMATE.....	456
9.12	TOOLS ON/OFF.....	457
9.13	SUMMARY.....	457
<b>OPTION MENU.....</b>		<b>459</b>
10.1	MESH CONTROL.....	460
10.2	FILE OPTION.....	462
10.3	SYSTEM LANGUAGE.....	464
10.4	MATERIAL LIBRARY STANDARD.....	465
10.5	LEFT MENU.....	465
10.6	SHOW ICON TIPS (TOGGLE).....	466
<b>UTILITIES.....</b>		<b>467</b>
11.1	ANGLE BETWEEN LINES.....	468
11.2	DISTANCE BETWEEN POINTS/NODES.....	468
11.3	RADIUS THROUGH 3PTS/3NDS.....	468
11.4	AREA OF SELECTED ELEMENTS.....	468
11.5	LENGTH OF LINE.....	469
11.6	SECTION CUT.....	469
11.7	DRAW ARROW.....	471
11.8	DEFINE TITLE.....	472
11.9	IDENTIFY NODE/POINT.....	472
11.10	IDENTIFY ELEMENT.....	473
11.11	FIND ELEMENT.....	473
11.12	FIND NODE.....	473
11.13	LOAD CURVE.....	473
11.14	SHOW LINE.....	489
11.15	COORDINATE SYSTEM.....	489
11.16	DATABASE STATISTIC.....	494
<b>VIEW OPTIONS.....</b>		<b>496</b>
12.1	COLOR MAP.....	496
12.2	ROTATION.....	498
12.3	LIGHT.....	499
12.4	DEFINE ROTATION CENTER.....	499
12.5	USER VIEW.....	500
12.6	TRUE VIEW.....	501
12.7	LABEL NODES (TOGGLE).....	501
12.8	LABEL ELEMENTS (TOGGLE).....	502
12.9	PLATE NORMAL (COLOR).....	502
12.10	FADE BACKGROUND.....	502

12.11 SHADING MODE .....	503
12.12 SHADING QUALITY .....	503
12.13 SCALE ACTIVE WINDOW .....	503
<b>ANALYSIS</b> .....	<b>504</b>
13.1 LS-DYNA (CTRL+A) .....	505
13.2 LS-DYNA (SPRINGBACK) .....	514
13.3 LS-DYNA (GRAVITY LOADING) .....	515
13.4 MSTEP .....	516
13.5 OUTPUT NEW DYNAIN FILE .....	516
<b>APPENDIX A</b> .....	<b>517</b>
<b>APPENDIX B</b> .....	<b>518</b>
<b>APPENDIX C</b> .....	<b>519</b>
<b>APPENDIX D</b> .....	<b>520</b>
<b>FINAL NOTES</b> .....	<b>521</b>

## INTRODUCTION

*eta/DYNAFORM Version 5.5* software is an *LS-DYNA-based sheet metal forming simulation solution package* developed by Engineering Technology Associates, Inc. The specialty CAE software combines the analysis power of LS-DYNA Versions 960 and 970 with the streamlined pre- and post-processor functions. These analysis codes and interactive functions are uniquely integrated to service the sheet metal forming industry in tooling design and development. The program also maximizes traditional CAE techniques to reduce prototyping costs and lead time for product development.

eta/DYNAFORM's analysis engine is LS-DYNA, which is developed and currently supported by the Livermore Software Technology Corporation (LSTC) of Livermore, California. It is a general purpose, non-linear, dynamic, finite element analysis code utilizing explicit and implicit capabilities for solving fluid and solid structural problems. The code has been developed for applications such as automobile crashworthiness, occupant safety, underwater explosion, and sheet metal forming.

The bottleneck of the metal forming development cycle is the hard tooling design lead-time. The eta/DYNAFORM CAE approach simulates this tooling process and thereby reduces the tooling tryout time and the associated cost for producing high quality stamped parts. eta/DYNAFORM effectively simulates the four major design concerns in the tooling process: *Binder Wrap*, *Draw Die*, *Spring Back*, and *Multiple Stage Tooling*. These simulations enable engineers to conduct feasibility studies of a product design early in the design cycle.

eta/DYNAFORM features well-defined tooling surface data to predict the performance of a stamping for areas such as *tearing*, *wrinkling*, and *thinning*, in addition to predicting *skid mark* and *spring back* effects.

### **Availability:**

eta/DYNAFORM is available for all UNIX workstation platforms including DEC(Alpha), HP, IBM, SUN, and SGI. It is available on PC for Windows and NT operating systems of 1998 and above. For LINUX, the Red Hat version 7.2 and above is supported.

### **Special features of eta/DYNAFORM Version 5.5 include:**

#### **Add new Setup module- AUTOSETUP**

This is a brand new module in the eta/DYNAFORM 5.5. This module is process oriented and enables user to easily setup the simulation parameters. Currently, both Sheet Metal Forming and Tubular Hydroforming interfaces are implemented in the Auto-Setup. In the Sheet Metal Forming interface, the local system coordinate can be defined so that the local Z axis can be

set as the stamping direction. Moreover, the tailor welded and the laminated blanks are supported.

### **Update Drawbead function**

The Drawbead function is updated based on the practical need. Compared with the drawbead function in former version, the latest drawbead function is improved in all kind of aspects. In previous version, it is very inconvenient for user to define and edit the drawbead. In eta/DYNAFORM 5.5, the GUI of drawbead and operation style is overhauled to facilitate easy drawbead definition and modification. All the operation in drawbead, including property definition, create, edit and restrain force definition, can be easily done without advanced training.

### **Improved Multiple Parts Nesting**

In eta/DYNAFORM5.5, the arithmetic of Multiple Parts Nesting function is improved and optimized. The result of auto nesting for multiple parts is more practical than former version. At the same time the material utilization is also enhanced.

### **More functional DFE module**

Based on the eta/DYNAFORM5.2, some functions of the DFE module in eta/DYNAFORM 5.5 are improved and added to satisfy the need of the friendly and functional operation. The major improvement includes:

- Added the TRIM LINE function. This function enables user to unfold the flange to the addendum created in eta/DYNAFORM and provide the accurate trim line for the subsequent trim operation.
- Added the RE-ENGINEERING module. This function enables user to convert the addendum which was designed by CAD software to the addendum in eta/DYNAFORM's DFE module. User can modify the addendum with tools provided in eta/DYNAFORM's DFE environment. Another key function of the Re-Engineering is the migration of addendum design from a part to another part which has minor shape change. Such capability reduces time in draw die design.
- The INNER FILL function in DFE module is drastically improved. User can fill both inner and boundary hole through defining some auxiliary lines. This function is very useful for user to fill the large hole, such as the window area of doors, lift gates, etc.
- In TIPPING and OUTER SMOOTH functions, user can select different part to perform Tipping and Smooth function.

### **Enhanced Pre-processor functions**

- Added the Delete function in the Part menu to allow removal of more than one part each time the function is activated.
- Improved Combine Line function.
- The Line Offset function is improved. A closed line will be created after offsetting a selected closed line.
- Improved line Re-space function.
- Improved the Element Split function. The \*Constraint Adaptive keyword will be generated after divided the elements of stamped part.
- Added the Filter option in the Select Element. The user can select one type of element by filtering element type.
- The Mid-Surface function is improved. The user can easily generate the mid-surface of a 3D solid part.
- The content of index file is updated to satisfy the need of user. Some basic information defined in preprocessor can be transferred to the eta/POST.
- The Model Check and Model Repair function is combined into one dialog box.
- The Guide/Pin function is added in the Tool menu. This function is developed to create the column or diamond-like Guide/Pin.
- Added the D3plot Curve function in the Utilities/Load Curve menu. This function enables user to set the time or displacement to output the d3plot file during the simulation.
- Added the POP Line function. User can define the POP Line in preprocessor, and then the POP line information will be transferred to the eta/POST through index file.

#### **Updated DYNA input file to support LS-DYNA 970 (6367 version)**

The LS-DYNA input file created by eta/DYNAFORM 5.5 supports LS-DYNA 970 (6367 version).

#### **Documentation**

A **Training Manual** is available to assist first time users with the general operating procedures of the program. The manual includes a relatively simple demonstration of a draw die simulation of an S-channel. The step-by-step procedure guides users through the complete process and flow of tasks. A new user can usually complete the training program and be ready to begin a production project in less than two hours.

The **eta/DYNAFORM Applications Manual** discusses the basics of the sheet metal forming process and transmits some fundamental knowledge of tooling design and related topics to the user. The following are typical examples of simulations that are included to demonstrate eta/DYNAFORM's capabilities, features, and functions:

- Case 1. NUMISHEET'2002 Fender Draw Simulation
- Case 2. Double Action of Hemisphere Draw Simulation
- Case 3. T-Shape Tubular Hydroforming Simulation

- Case 4. NUMISHEET'2002 Fender Gravity Simulation
- Case 5. Tailor welded blank and Multi-Stages Simulation
- Case 6. U-Channel Springback Simulation
- Case 7. Cup Sheet Hydroforming Simulation

These workshop examples enable users to fully understand general pre- and post-processing operations in addition to the applications of binder wrap simulation, draw bead force calculation, draw die simulation, Forming Limit Diagram (FLD), thickness, thinning, wrinkling, and springback simulation.

The eta/DYNAFORM **User's Manual** contains function descriptions specifically to the eta/DYNAFORM interface.

For further details on LS-DYNA, please refer to the manuals of the codes published by the LSTC.

# Chapter 1

## STRUCTURAL OVERVIEW

### 1.1 GENERAL

eta/DYNAFORM Version 5.5 is a complete Graphic User Interface (GUI) package operating on Windows (NT, 98 and above) and on UNIX/Linux based workstations (W/S) including IBM, HP, DEC-Alpha, SGI and SUN-SOLARIS platforms. The model generation and input file preparation of a typical stamping simulation are performed in the eta/DYNAFORM preprocessor, while the stamping simulation results is processed in eta/POST. Please refer to the eta/POST User's Manual for detailed information. The solver, LS-DYNA can be executed on either local or remote server systems.

eta/DYNAFORM is organized as a tree structure and is operated and controlled by user-friendly GUI. The environments are unified on both Windows and UNIX platforms.

### 1.2 MENU BAR

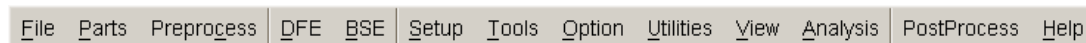


Figure 1.1 eta/DYNAFORM menu

As illustrated in Figure 1.1, eta/DYNAFORM menus are selected by mouse pick and contain a wide variety of functions. Descriptions for these functions are located in their respective sections.

FILE MANAGER	Import/exports data to/from eta/DYNAFORM
PART CONTROL	Organizes Parts (or layers)
PREPROCESS	Contains the menu of preprocessing functions such as lines/points, elements, nodes, check functions, boundary conditions
DFE	DIE FACE ENGINEERING provides tools to quickly derive a suite of die which includes addendum and binder
BSE	BLANK SIZE ENGINEERING includes a modified one-step-based unfolding function, blank development and nesting function
SETUP	The SETUP menu provides a streamlined process for setting up standard and advanced stamping simulations
TOOLS	Provides functions to create, define, and modify tooling and blank definitions.
OPTION MENU	Contains various options to control the mesh criteria, file window style, material library and language setup

UTILITIES	Includes several supplemental functions for identifying entities, measurement and load curve generation
VIEW OPTIONS	Controls the display of the items and view angle on the screen
ANALYSIS	Defines the control parameters, outputs the DYNA input deck, and submitting job
POSTPROCESS	Initiates eta/POST for post-processing analysis results
HELP	Displays the version level and contact information for eta/DYNAFORM support and Online Help (F1)

### 1.3 ICON BAR

The icon bar is designed to provide the user easy access to the most commonly used functions of eta/DYNAFORM. The user can click on these icons to activate the functions instead of browsing the various menus. An illustration of icon bar is shown in Figure 1.2.



Figure 1.2 Icon bar



#### **NEW**

Creates a new database file.



#### **OPEN**

Opens or finds a database.



#### **IMPORT**

Imports CAD and FE model files, such as IGES, VDA, DYNA, etc into the current database.



#### **SAVE**

Updates the current database.



#### **PRINT**

Creates a postscript file of the display area, and sends the file to the printer (default) or to a file. Prior to printing, the postscript driver must be initialized to accommodate eta/DYNAFORM software.



#### **TURN PARTS ON/OFF**



Turns the selected parts ON or OFF. The PART TURN ON/OFF dialog window is displayed to select parts.

**DELETE ALL UNREFERENCED NODES**

Deletes all nodes that there are not associated to elements.

**VIRTUAL X ROTATION**

Dynamically rotates the displayed model about the global X-axis when the cursor is moved up or down.

**VIRTUAL Y ROTATION**

Dynamically rotates the displayed model about the global Y-axis when the cursor is moved up or down.

**VIRTUAL Z ROTATION**

Dynamically rotates the displayed model about the global Z-axis when the cursor is moved up or down.

**SCREEN X ROTATION**

Dynamically rotates the displayed model about the screen X-axis when the cursor is moved up or down.

**SCREEN Y ROTATION**

Dynamically rotates the displayed model about the screen Y-axis when the cursor is moved up or down.

**SCREEN Z ROTATION**

Dynamically rotates the displayed model about the screen Z-axis when the cursor is moved up or down.

**FREE ROTATION**

This function is a combination of rotation about screen X-axis (SX) and screen Y-axis (SY). To manipulate SX, move the mouse cursor up or down. To manipulate SY, move the mouse cursor left or right. Moving the mouse cursor diagonally combines both actions. To dismiss the function, click once at the left mouse button. This function can be initiated by simultaneously pressing Ctrl key and clicking left mouse button, following by moving the mouse cursor.

**PAN**

This command enables the user to translate the model by following the movement of the mouse cursor. If the cursor is moved off the screen, it will be reset at the center of the display screen. To dismiss the function, click once at the left mouse button. This function can be initiated by simultaneously pressing Ctrl key and clicking left mouse button, following by moving the mouse cursor.

**CURSOR ZOOM**

The function enables the user to reduce or enlarge of a model or selected region on the display screen. To reduce or enlarge a selected region, position the mouse cursor at the region. Then, click and hold left mouse button, following by moving the mouse cursor up or down. This function can be initiated by simultaneously pressing Ctrl key and clicking right mouse button, following by moving the mouse cursor up or down.

**WINDOW ZOOM**

This function enables the user to enlarge a selection region of a model on the display screen. The user defines the corners of the zoom window by positioning the cursor on the display screen. Then, clicks the left mouse button and drags the cursor diagonally down to obtained desired window size. The entities included in the zoom window will be fully displayed on the screen after the left mouse button is released.

**FREE HAND ZOOM**

This function is similar to window zoom. Instead of using drag window, the user defines a free region by pressing the left mouse button and dragging the mouse cursor on the display screen. The entities included in the circled zone will be fully displayed on the screen after the left mouse button is released.

**ACTIVE WINDOW**

This command allows the user to isolate a portion of the displayed geometry/model for more detailed viewing or editing. The user defines the region by dragging a window on the desired area. eta/DYNAFORM displays the elements, lines, and surfaces within the window (volume in space) as active. Other objects on the screen remain masked and inactive.

**FILL**

Rescales and fits all the entities of the model on the display screen.

**TOP VIEW**

Displays the model from the TOP or in the XY-plane.

**LEFT VIEW**

Displays the model from the LEFT or in the XZ-plane.

**RIGHT VIEW**

Displays the model from the RIGHT or in the YZ-plane.

**ISOMETRIC VIEW**

Displays the model from the isometric plane (120-degree isometric).

**CLEAR**

Remove the highlighted entities from the screen, such as those made with the commands, SHOW LINE, BOUNDARY CHECK, ID ELEMENTS, and DEFINE TITLE.

**REDRAW**

eta/DYNAFORM updates the screen after each command. Occasionally, the use of specific commands requires updating of the image with REDRAW function (e.g., when operating the dynamic zoom at the same time that the element normal are displayed, it is necessary to activate the REDRAW command to resize the arrows that shows the normal direction of elements).

**UNDO**

Undo the last operation. If no operation is recorded, the icon is disabled.

**REDO**

Redo the last operation. If no operation is recorded, the icon is disabled.

## 1.4 DISPLAY WINDOW

eta/DYNAFORM divides the screen into six distinct regions. As illustrated in Figure 1.3, the regions are used to receive input or list messages. The six regions are described below:

1. DISPLAY AREA  
Show models and graphs.

2. MENU BAR  
Display commands and command options.
3. ICON BAR  
Provide the user easy access to the most commonly used functions of eta/DYNAFORM.
4. DIALOG WINDOW DISPLAY  
Once the user selects a command from the MENU BAR, a corresponding dialog window with the appropriate functions is displayed in this area.
5. DISPLAY OPTIONS  
This group of commands is always displayed and can be used at any time during an eta/DYNAFORM session.
6. PROMPT AREA  
Display the eta/DYNAFORM comments and messages.

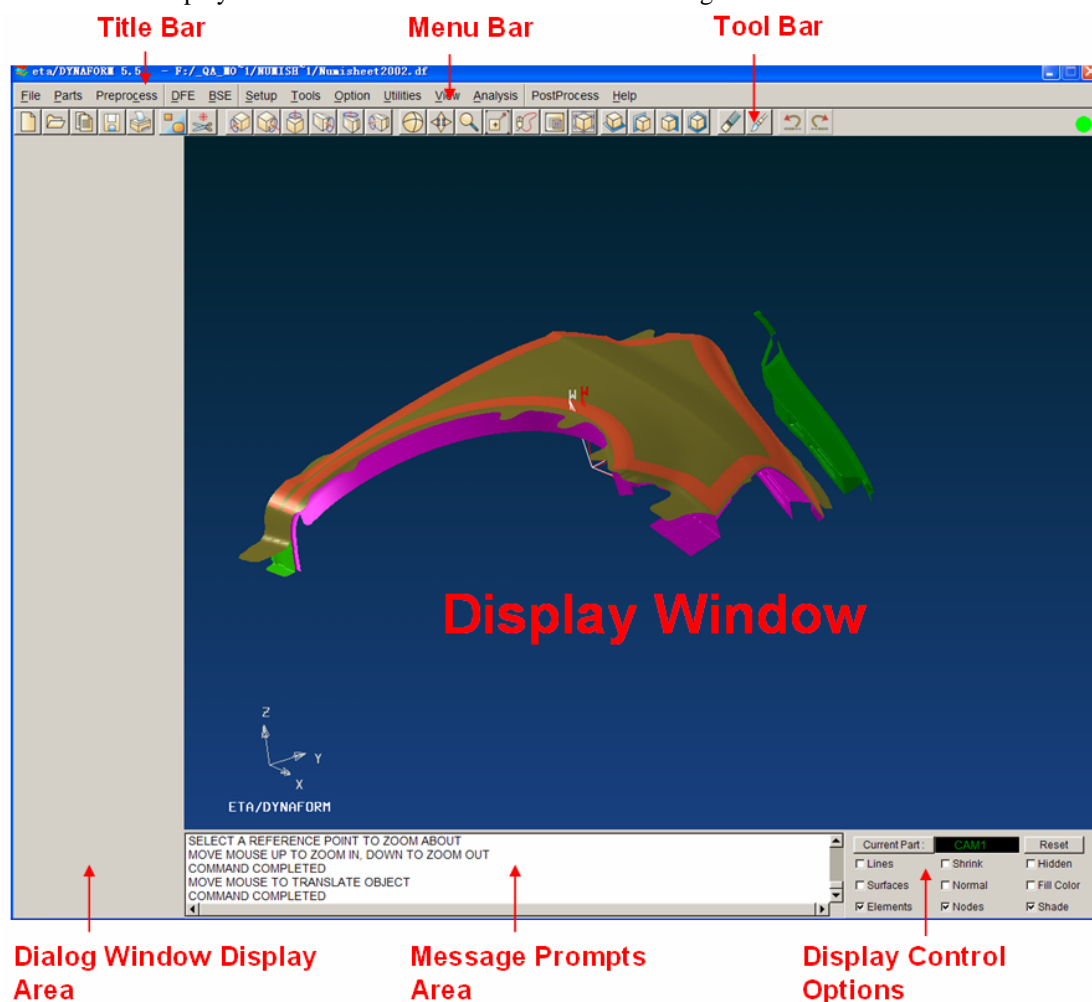


Figure 1.3 Interface of eta/DYNAFORM

## 1.5 DISPLAY OPTIONS

The DISPLAY OPTIONS window, as illustrated in Figure 1.4, is located at the bottom right corner of the display screen. It displays the current part and contains the following commonly used functions.

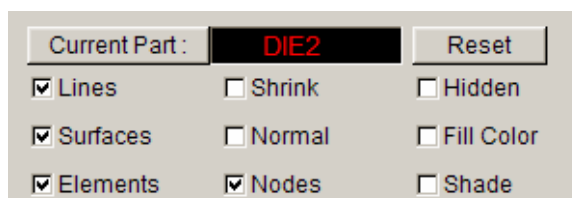


Figure 1.4 Display Options

### CURRENT PART

This function enables the user to set the current part. It is very important to set the current part in the eta/DYNAFORM because most of the results generated from the different functions will be stored into the current part by default. The user can click on the Current Part button to change the current part. After click the button, the system will pop up a Current Part dialog box, as illustrated in Figure 1.5. The user can choose one of the parts as the current part. The current part name will be displayed in the text box to the right of the Current Part button.

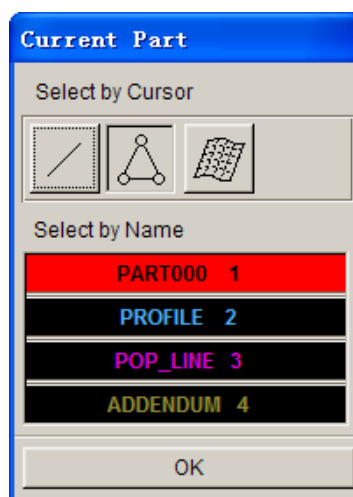


Figure 1.5 Current part dialog box

### RESET

Restore all options to the default.

### LINE (toggle)

Toggle checkbox on/off to show/hide the lines on display window.

### SURFACE (toggle)

Toggle checkbox on/off to show/hide the surfaces on display window.

**ELEMENT (toggle)**

Toggle checkbox on/off to show/hide the elements on display window.

**SHRINK (toggle)**

Create a plot with element size being reduced by 20%. As shown in Figure 1.6, SHRINK ELEMENTS is useful for locating any missing and/or degenerated elements in a shell or solid structure.



Figure 1.6 Shrink Elements

**NORMAL (toggle)**

This function enables the user to display the normal direction of an element with an arrow at the center of the element and perpendicular to the surface of the element. For a solid element, the arrow points toward the bottom surface of the element.

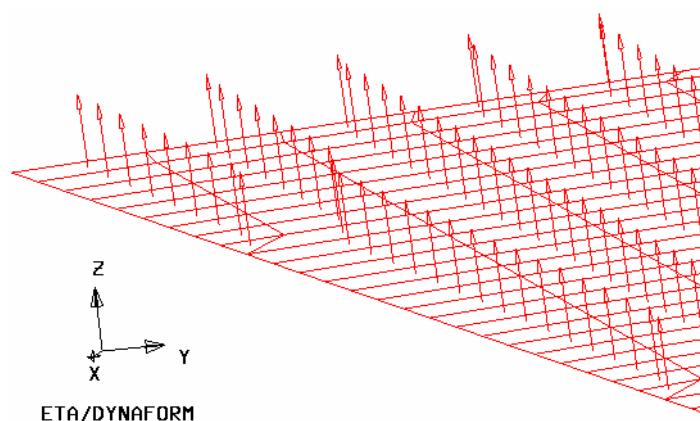


Figure 1.7 Elements Normal

If the user toggles on the PLATE NORMAL (COLOR) in the VIEW menu, the direction of element normal will be displayed using part color. See Figure 1.8.

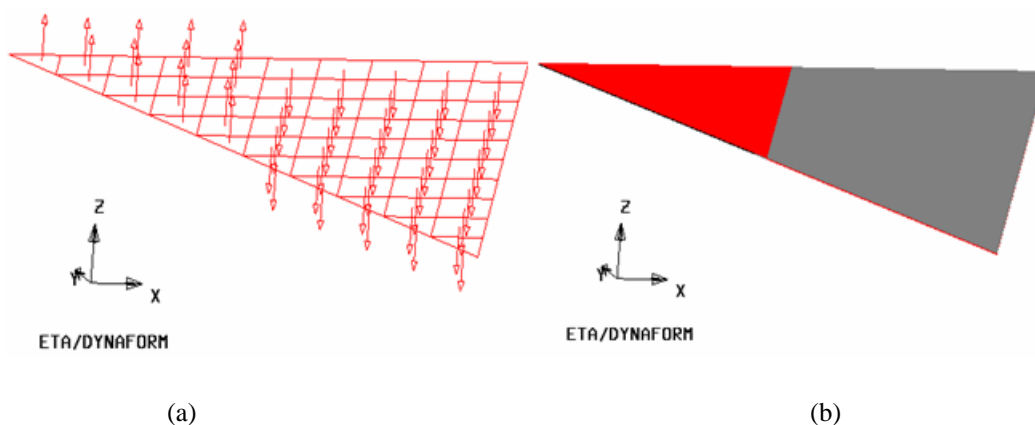


Figure 1.8 Display of elements normal direction (a) with arrows (b) with colors

#### **NODE (toggle)**

Toggle on/off checkbox to show/hide nodes.

#### **HIDDEN (toggle)**

This function improves the display integrity of 3D model. The user may toggle on/off checkbox of the hidden lines function while the SHADING option is toggled on. This effect creates opaque elements that prevent objects in the background from showing through objects in the foreground.

#### **FILL COLOR (toggle)**

The FILL COLOR function fills displayed elements with a designated color. The function is unable to accurately represent the depth perspective of a model. The displayed part may appear to warp or penetrate with each other. However, the FILL COLOR command displays an accurate 3D model display of the parts when it is used in conjunction with the HIDE PLOT option.

#### **SHADE (toggle)**

Display the 3D parts as if they were illuminated by a light source. Parts that are not directly exposed to the light source are appropriately shaded to imitate actual shading.

## 1.6 MOUSE FUNCTIONS

eta/DYNAFORM functions are accessible via the left mouse button. To access a function, the user moves the mouse cursor to select the desired function, following by depressing the left mouse button. In certain functions, the middle mouse button is used to complete operations, such as creating a line and selecting nodes.

## 1.7 SPECIFICATIONS

The standard version of eta/DYNAFORM has the following specifications for PC and UNIX/LINUX based workstations per database:

150,000	LINES
900,000	POINTS
8,000	SURFACES
600,000	EDGE POINTS (SURFACE)
800,000	CONTROL POINTS (SURFACE)
16,000	BOUNDARIES (SURFACE)
500,000	ELEMENTS
500,000	NODES
1,000	PROPERTIES
1,000	MATERIALS
1,000	PIDS
1,000	LOCAL COORDINATE SYSTEMS
8	LENGTH OF PART, MATERIAL, PROPERTY NAMES

The large version of eta/DYNAFORM has the following specifications for PC and UNIX/LINUX based workstations per database:

64,000	SURFACES
12,000,000	EDGE POINTS (SURFACE)
64,000,000	CONTROL POINTS (SURFACE)
2,000,000	ELEMENTS
2,000,000	NODES

## 1.8 GEOMETRY DATA

eta/DYNAFORM directly reads in IGES, VDA, STEP and eta/DYNAFORM/FEMB geometry data (line and surface). It also directly reads in CATIA4 & 5, Pro-E and UG part files with native libraries.

## 1.9 RECOMMENDED NAMING CONVENTION

The protocol for naming files during an eta/DYNAFORM session includes attaching suffixes that specify the file types to the file names. The appropriate file names are listed in the options area of the display screen.



Examples of suffixes include:

eta/DYNAFORM database file name:	<i>filename.df</i>
eta/DYNAFORM geometry data file name:	<i>filename.lin</i>
IGES geometry data file name:	<i>filename.igs, or filename.iges</i>
VDA surfaces data file name:	<i>filename.vda, or filename.vdas</i>
Data Exchange file name:	<i>filename.dxf</i>
Stereo lithography file name:	<i>filename.stl</i>
ACIS file name:	<i>filename.sat</i>
CATIA4 database file name:	<i>filename.model</i>
CATIA5 database file name:	<i>filename.CATPart</i>
STEP file name:	<i>filename.stp</i>
UG database file name:	<i>filename.prt</i>
Pro-E database file name:	<i>filename.prt or filename.asm</i>
NASTRAN input file name:	<i>filename.nas or filename.dat</i>
LS-DYNA input file name:	<i>filename.dyn or filename.k</i>
LS-DYNA model file name:	<i>filename.mod</i>
LS-DYNA input file name:	<i>filename.k</i>
LS-DYNA input file name:	<i>dynain</i>
ABAQUS input file name:	<i>filename.inp</i>

Example: When reading in a line data file, eta/DYNAFORM prompts for a line data filename (all file names in that directory with the suffix **.lin** are listed in the options area). The user then selects the appropriate file name. If the user wishes to import multiple files in the same directory, he/she can click on the IMPORT button after the file name is selected. The dialog box will stay open until **OK** or **CANCEL** is clicked.

This practice simplifies file name selection and organizes the user's work directory.

## 1.10 DIALOG BOXES

eta/DYNAFORM incorporates various dialog boxes to execute functions throughout the program. At the bottom of the dialog boxes there are buttons to execute, reject, reset the data or close the dialog box. The functions of these buttons are listed below.

<b>ABORT</b>	Aborts the current function and exits the dialog box.
<b>APPLY</b>	Executes the current function without leaving the dialog box.
<b>BACK</b>	Returns the user to the previous dialog box.
<b>CANCEL</b>	Rejects the current operation.
<b>CLOSE</b>	Closes the current dialog box.

<b>DONE</b>	Finishes the current step in the dialog box and allows the user to proceed to the next step.
<b>EXIT</b>	Exits the current dialog box.
<b>OK</b>	Accepts the data in the dialog box and forwards the user to the next step.
<b>UNDO</b>	Rejects the last step of the operation.
<b>REJECT</b>	Rejects the previous selection.

## 1.11 PROPERTY TABLES

The tables in eta/DYNAFORM allow the user to modify properties of materials, elements, drawbeads, etc. For example, the material window as shown in Figure 1.9 enables user to edit material properties such as K, N, R00, R45, R90, etc.

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 1.9 Material Type 36 Window

**OK** - Accept the currently displayed values and exits the table.

**ADVANCED** - Activate advanced parameters. Once the advanced parameters are activated, **ADVANCED** becomes **REGULAR**.

**DEFAULT** - Set all fields to their default values.

**RESET** - Reset the last altered field to its previous state.

**CANCEL** – Exit the table without entering any altered values.

The data fields in the table can be edited using several different ways: 1. Click the mouse button to insert the cursor at a specific point in the data field; 2. Click and hold the left mouse button, following by dragging the cursor to highlight a selected portion of the field; 3. Double click left mouse button to highlight the entire data field. Keystroke entry then inputs the new value. New values will be recorded in the table. Most parameters are organized into two groups: standard and advanced. Only the standard parameters are displayed on the window. At the bottom of the table the OK, ADVANCED, DEFAULT, RESET, and CANCEL buttons allow the user to accept or reject the data. The ADVANCED button is implemented in the following dialogue boxes: MATERIAL, PROPERTY, CONTACT PARAMETERS, CONTROL PARAMETERS, DRAWBEAD PROPERTIES and ADAPTIVE PARAMETERS.

## 1.12 CONFIGURATION FILE

Three configuration files (**.dynaformDefault**, **dynaformhardcopydefault** and **dynaform.ini**) are created to set all the control parameters. By default, these files are located in the “My Documents” directory of the current user’s PC. The files are located in the user's home directory on UNIX/LINUX. The modification of these files may require super-user privilege.

**DynaformDefault** primarily controls the default settings of analysis parameters. Each parameter has a control flag to control permission for modification. The format of each control parameter is listed as the following:

“**Keyword value permission key**”.

The user can modify the default settings of most parameters related to a stamping simulation.

**Dynaformhardcopydefault** controls the default print settings. The format of each control parameter is listed as the following:

“**Keyword value**”

The user can modify the default settings of most printing parameters.

**dynaform.ini** is provided to save customized settings in the database or in the user’s home directory. The format of each parameter is listed as the following:

```
#DYNAFORM5.5 configure file
#[keywords] value
#[FILE_DIALOG_STYLE]: 0 - WINDOWS style, 1 - UNIX STYLE
```

```

#[SYSTEM_LANGUAGE]: 0 - ENGLISH, 1 - CHINESE, 2 - JAPANESE, 3 - KOREAN
#[AUTOBACKUPTIME]: <=0 - don't autobackup, >0 - backup time( minutes )
#[AUTORESTART]: 0 - don't autorestart, 1 - autorestart.
[FILE_DIALOG_STYLE]
0
[SYSTEM_LANGUAGE]
0
[LAST_DATABASE_PATH]
D:/DYNAFORM_MATERIAL/DFE_Tutorial_manual/Case4_HoodInner/
[LAST_IMPORT_PATH]
D:/dynaform55/dir_dfe/
[LS-DYNA_SOLVER]
D:\DYNAFO~5\lsdyna.exe
[AUTOBACKUPFILE]
timed.df
[AUTOBACKUPTIME]
0
[AUTORESTART]
0

```

Comment lines start with '#'.

**[FILE\_DIALOG\_STYLE]** Determine the default style of file dialog. There are two file dialog styles (Windows and UNIX) under the Option/File option.

**[SYSTEM\_LANGUAGE]** Define the default language for the GUI. eta/DYNAFORM currently supports four types of languages. There are: Chinese, English, Japanese and Korea

**[LAST\_DATABASE\_PATH]** When saving or opening a database, eta/DYNAFORM will automatically identify the last saved or opened database path.

**[LAST\_IMPORT\_PATH]** When importing or exporting a database, eta/DYNAFORM will identify the last import or export database path automatically.

**[LS-DYNA\_SOLVER]** When running a model, eta/DYNAFORM will search for the LS-DYNA solver from this path.

**[AUTOBACKUPFILE]** Define the default backup file name.

**[AUTOBACKUPTIME]** Define the time interval between automatic database saving.

**[AUTORESTART]** Postprocess can be initiated directly from eta/DYNAFORM. If the value is 1, the database (\*.df file) will be restarted automatically after eta/POST interface is closed.

## Chapter 2

# GETTING STARTED

eta/DYNAFORM is organized into three major modules, a formability simulation module, blank size engineering module, and a die face engineering (DFE) modules. All modules can be executed independently from eta/DYNAFORM interface.

### 2.1 GETTING STARTED

An eta/DYNAFORM session begins with one of the following scenarios:

1. The user provides CAD-IGES or VDA or any other geometry data that will be translated and read directly into eta/DYNAFORM.
2. The user provides model data (e.g. NASTRAN, LS-DYNA, etc.) that will be read directly into eta/DYNAFORM.
3. CAD or model data is not provided. The user begins with a blank database and generates or digitizes the line and surface data from a drawing.

*NOTE: An example of a method of supplying or generating data is outlined in Sections 2.2~2.4.*

Usually, a stamping simulation includes the following steps.

### 2.2 STARTING WITH CAD DATA

The user begins a session by entering eta/DYNAFORM executable, such as “dynaform”, “df”, etc., at the UNIX prompt. For Window/LINUX, the user can double clicking on the eta/DYNAFORM icon from the desktop or select eta/DYNAFORM from program menu. Once the program is activated, the default database named Untitled.df is created.

#### 2.2.1 STARTING WITH CAD SURFACE DATA

The user can read in surface data via the command FILE/IMPORT. The user can choose between IGES, VDA, Line Data or any other geometry file types.

**NOTE:** If *IGES* is selected, files having the suffixes *.igs* and *.iges* will be listed in the menu area. Please refer to Section 1.8 to get more information on file types.

If *LINE DATA* is selected, files having the suffix *.lin* will be listed.

### 2.2.2 STARTING WITH CAD LINE DATA

By using the line data, the user can create models via the *PREPROCESS/ELEMENT OPTIONS* menu. The functions in this menu can be used to create a 2-, 3-, or 4-LINE MESH directly from the line data. In addition, lines can be used to generate surfaces using the *SURFACE* option menu located in the *PREPROCESS/SURFACE* menu. The created surfaces can be auto meshed via the *SURFACE MESH* command found in the *ELEMENT* option menu. The line data can be imported to *eta/DYNAFORM* database using the *FILE/IMPORT* command.

## 2.3 GETTING STARTED WITH MODEL DATA (NASTRAN, LS-DYNA, etc.)

1. To read an *LS-DYNA* (or other analysis code) data file into *eta/DYNAFORM* database, select *FILE/IMPORT*.
2. A dialog window will appear. Select the drop down menu to the right of *Files of Type* and choose the desired file format, following by selecting the desired file from the dialog window.
3. Once the selection has been made, the program will read the *LS-DYNA* input file into *eta/DYNAFORM*.
4. At this point, the display can be manipulated via the commands in the *ICON BAR* and the *DISPLAY OPTIONS* area.

## 2.4 GETTING STARTED FROM SCRATCH

This section will help users getting started in *eta/DYNAFORM* when line or model data is unavailable. The user will typically generate his/her own line data according to the following procedure:

1. Create a new part in the *PARTS* menu.
  - Lines or elements cannot be created in an empty database. The user must create a part before defining any geometry of model data.

2. Generate line data by its coordinates. Refer to Section 5.1.1 in Chapter 5, PREPROCESS, for additional information on generating lines.
  - Exit the PARTS menu.
  - Enter the PREPROCESSING/LINE/POINTS menu.
  - Select the CREATE LINE command.
  - Select the LCS or ABS option for the coordinates in the current local or global coordinate system. Refer to Section 11.14, LOCAL COORDINATE SYSTEM.
  - Select the XYZ or DXYZ option.
  - If XYZ is selected, the key-in values of U, V, and W are the coordinates.
  - If DXYZ is selected, the key-in values of DU, DV, and DW are increments of the previously created point.
  - Key in the three values in the corresponding field.
  - Select OK to confirm the values and create the point.

The prompt asks for the coordinates until the user selects either EXIT or DONE from the options area of the display screen.

eta/DYNAFORM lines consist of multiple points. Using the DONE command will create an end to a current line.

***NOTE: The user may continue to generate line data or execute the other CAD functions available in the PREPROCESS menu. Generating a model from line data is described in Section 4.3, ELEMENT.***

Once the desired geometry has been created and meshed, the user can define each part as a specific tool (e.g. a die, punch, upper/lower ring, or blank) via the TOOL DEFINITION menu. In this menu, the blank material and thickness and the tool positioning can also be defined. Once the tools are defined, the interface between the tools will automatically be generated during the writing of the LS-DYNA file.

A draw bead can be defined in the TOOL DEFINITION menu. Proceeding to ANALYSIS PARAMETER in the SETUP menu, the user can directly submit a job from the eta/DYNAFORM window via the ANALYSIS menu. Once the analysis is complete, the result files can be loaded and viewed by the POSTPROCESS portion of the program.

After the forming analysis is complete, the result file **dynain** can be read in to view the formed blank shape. This file is now ready for subsequent springback analysis or multi-step stamping simulation. A trimmed blank model can be written out as another dynain input file from the analysis output. A blank may be trimmed using the TRIM command under BLANK OPERATIONS submenu in the TOOL DEFINITION menu. Once the analysis is complete, the results can be viewed by reading the d3plot files into the POSTPROCESS portion of the program.

Please refer to Chapter 8.1 for detailed simulation procedure using QUICKSETUP.

Please refer to Chapter 6 for detailed steps for generation of die face using DFE.

## 2.5 LOCAL COORDINATE SYSTEM

eta/DYNAFORM refers to the local coordinate system to translate, rotate, mirror, copy, and generate points, lines, or nodes. When such a function is selected, the program will automatically prompt the user to generate a local system designated as the UVW coordinate frame. The LCS dialog box will be displayed (Figure 2.5.1). The user can create a new LCS or select a system as the current LCS by selecting GLOBAL, CURRENT LCS, LAST, or VIEW DIRECTION.

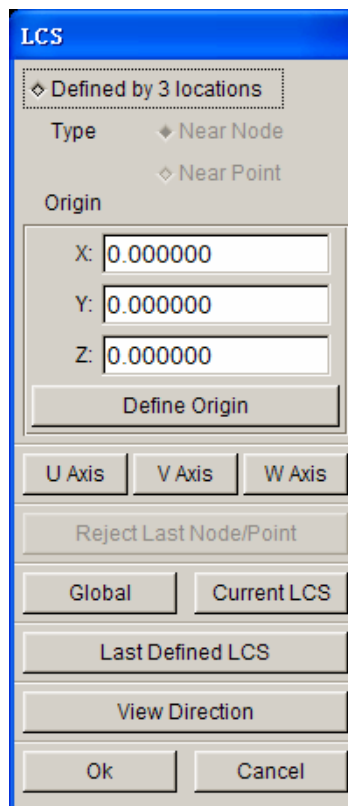


Figure 2.5.1 LCS dialog box



**GLOBAL**

Defines the global coordinate system as the current LCS.

**CURRENT LCS**

Creates a local coordinate system as the current LCS.

**LAST**

Defines the last coordinate system as the current LCS.

**VIEW DIRECTION**

Defines the view directions as the W-Axis of the current LCS. The user can select or define a point as the origin, or the origin of the global coordinate system can be the origin of the current LCS.

There are two ways to create a LCS based on whether the DEFINED BY 3 LOCATIONS toggle is checked.

**1. CHECKED (ON)**

The user can select one, two, or three points/nodes to create a system. To select a node, choose NEAR NODE. To select points, choose NEAR POINT.

**1. ONE POINT**

Select a point and choose OK. The created LCS has an origin at the selected point and is a translation to the origin of the global system.

**2. TWO POINTS**

Select two points and choose OK. The first point is the origin. The W-axis is the vector from the first point to the second point. The LCS is a translation and a rotation from X to U of the global system.

**3. THREE POINTS**

Select three points P1, P2, and P3. The point, P1, is the origin. The vector, P1-P2, is the U-axis. The cross product of U and the vector, P2-P3, is the W-axis. The V-axis is determined by the cross product of  $W \times U$ .

## 2. UNCHECKED (OFF)

DEFINE ORIGIN, U-AXIS, V-AXIS, and W-AXIS will be activated. To create a system in this case, the user needs to define an origin and axis. The LCS system will be a translation and a rotation of the global system.

- Select DEFINE ORIGIN, and the INPUT COORDINATE dialog will appear. Select a point or node as the origin, and then select OK.
- Select one of the AXIS buttons to define the axis in the LCS dialog.
- Select BY POINT or ALONG AXIS from the U/V/W AXIS dialog (Figure 2.5.2) to define the axis. If BY POINT is selected, select a point in the display window to define the vector for the desired axis. If ALONG AXIS is selected, the choices are ALONG X, ALONG Y, ALONG Z, ALONG U, ALONG V, ALONG W. After selecting an axis, select DONE.
- Select OK from the AXIS dialog.
- Select OK from the LCS dialog.

**NOTE:** *If the local coordinate system is not acceptable, use the NO button to reject and redefine the coordinate system.*

- The coordinate system is displayed.

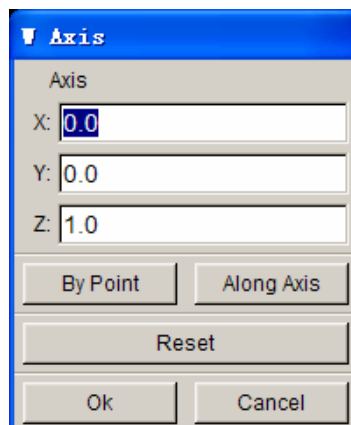


Figure 2.5.2 Define W Axis

## Chapter 3

# FILE MANAGER

The File Manager enables user to manage the active database, data input/output, submitting jobs and print setup. An illustration of the options in File Manager is shown in Figure 3.1.

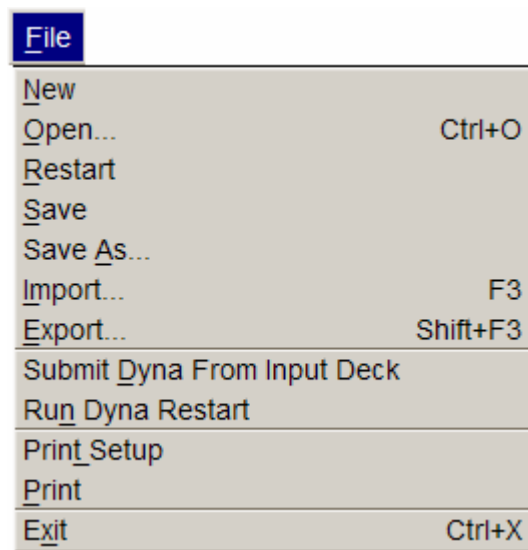


Figure 3.1 File Manager

A detailed description of each function is given in the following sections.

### 3.1 NEW

This function allows the user to create a new database file.

- When a database file is opened in eta/DYNAFORM, a warning message window as shown in Figure 3.3.1 prompts the user to save the active database file before creating a new one.

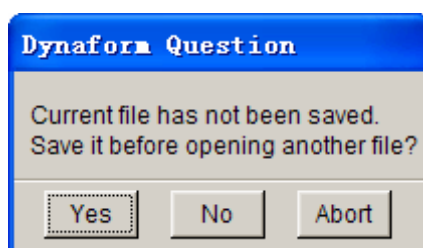


Figure 3.1.1 New File Prompt

- After saving the current database, a new, empty database, Untitled.df, is created automatically.

### 3.2 OPEN (CTRL+O)

This function allows the user to open a database.

- If the selected database is active, eta/DYNAFORM prompts the user to save the active file before opening the selected database.
- The user can open a eta/DYNAFORM database by selecting a file name in the open file dialog window. The OPEN DATABASE window is shown in Figure 3.2.1.

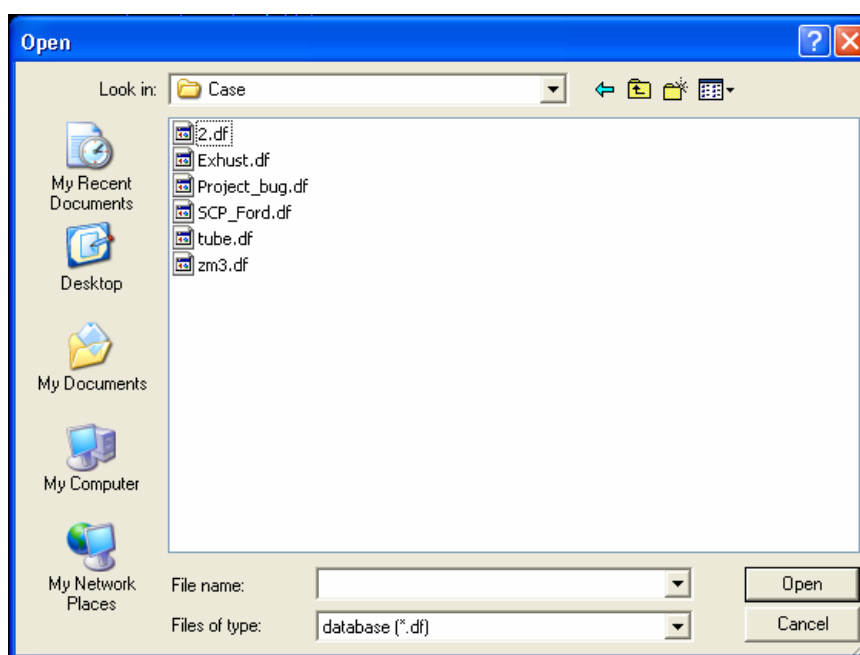


Figure 3.2.1 Open File window

### 3.3 RESTART

This function allows the user to restart the active database from the previously saved stage. eta/DYNAFORM prompts the user to save the active file prior to restarting the database. The message window is shown in Figure 3.3.1.

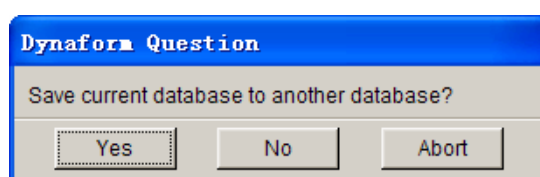


Figure 3.3.1 Dynaform Question message window

- Select YES to save the database with another name.
- Select NO and any unsaved data will be lost.
- Select ABORT to cancel the restart operation.

### 3.4 SAVE

This function saves the active database with its current name and location.

### 3.5 SAVE AS

This function saves the current database as a new file. The Save As window is shown in Figure 3.5.1.

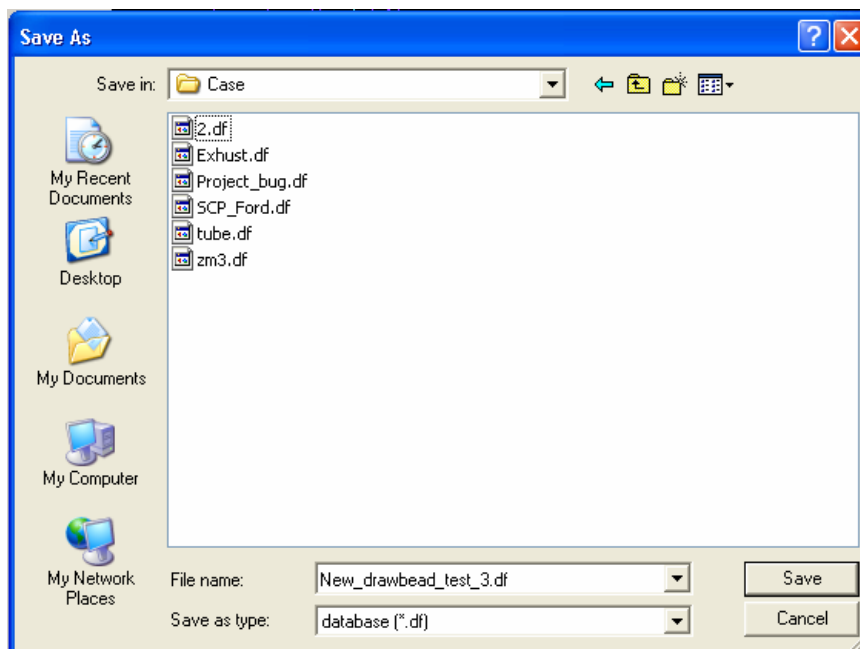


Figure 3.5.1 Save File As window

### 3.6 IMPORT (F3)

This function allows the user to read in CAD or finite element model data into active database. The Import window is shown in Figure 3.6.1.

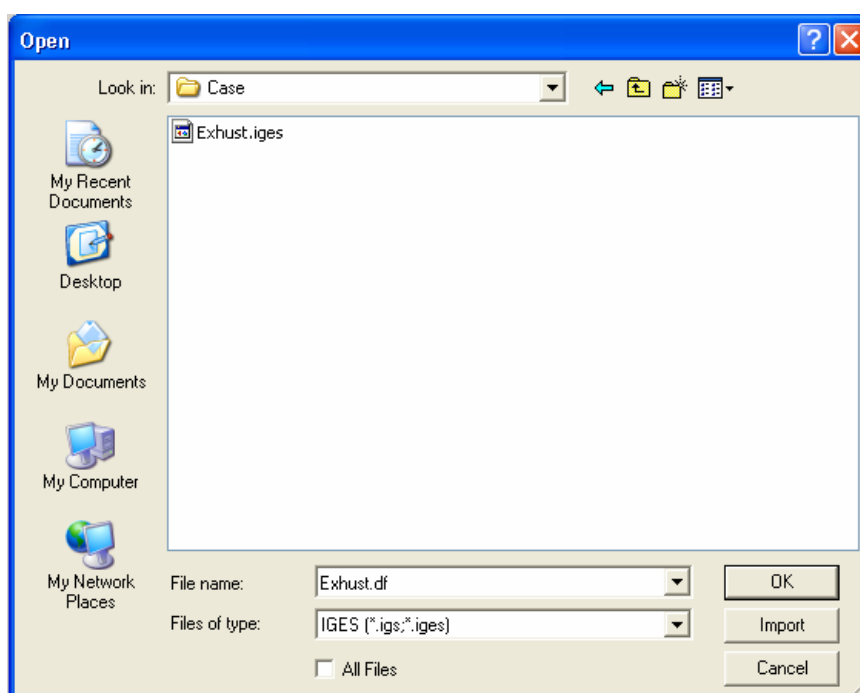


Figure 3.6.1 Import File window

**FILE TYPE**

Select the downward pointing triangle in the FILE TYPE dropdown menu and select the proper format. The available formats are:

MODEL DATA FORMAT: LS-DYNA, NASTRAN, and ABAQUS

CAD DATA FILE: LINE DATA, IGES, VDA, DXF, STL, ACIS, CATIA, STEP, UG, PRO-E, and

MULTI-STAGE FILE: DYNAIN

If ALL FILES is selected, all types of files will be shown in the window.

The IMPORT button allows the user to continue to import files without closing the IMPORT FILE window.

**3.7 EXPORT (SHIFT+F3)**

This function allows the user to output files from the active eta/DYNAFORM database. The options are similar to those mentioned above. The supported formats are:

**INDEX**

Includes basic model information from the eta/DYNAFORM database including the unit system, draw type and direction, tool IDs and names and material. This information will be used for

post-processing simulation results. The INDEX file will be written automatically when the DYNA input file is written.

### NASTRAN

Entire or partial model data.

### LINE DATA, IGES, VDA, ACIS, STEP, CATIA and UG PART

CAD data (lines and surfaces) of entire or partial geometry data.

## 3.8 SUBMIT DYNA FROM INPUT DECK

This function enables the user to submit the input file to LS-DYNA (solver) and automatically starts up the solver.

1. eta/DYNAFORM prompts the user to select the input file from the window as illustrated in Figure 3.8.1.

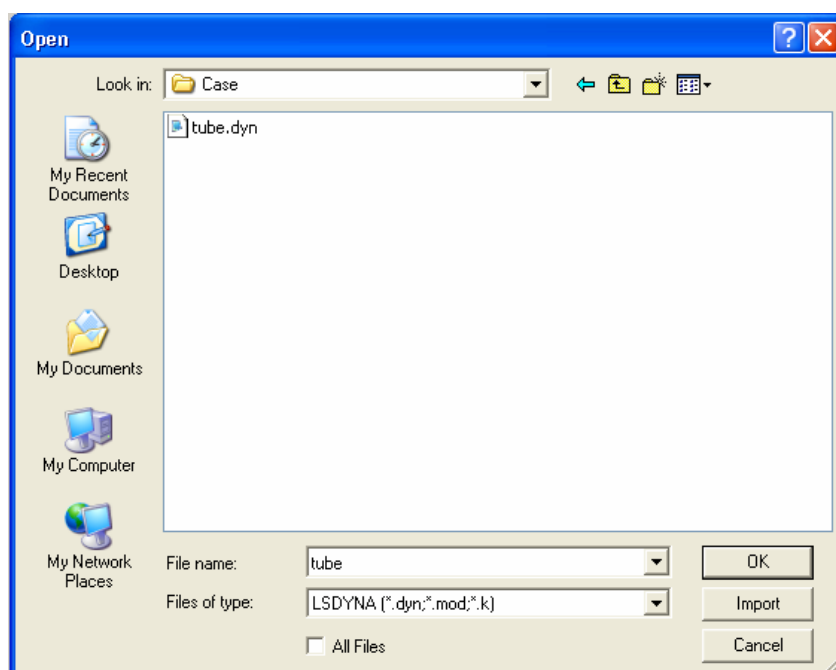


Figure 3.8.1 Select Input File window

The user can select the LSDYNA file or toggle on ALL FILES checkbox to list all files in the folder. Then, double click the left mouse button on the desired file or hit OK button to continue.



- The SUBMIT DYNA dialog window, as illustrated in Figure 3.8.2 appears. The dialog window enables user to specify the memory and type of LS- DYNA solver, either Single or Double precision. The default memory is set as 256MB. User can assign different memory value according to model size.

**OK** LS-DYNA starts and begins to solve the selected input deck file.

**BACK** Returns to the SELECT INPUT DECK FILE window.

**CANCEL** Exits the function.

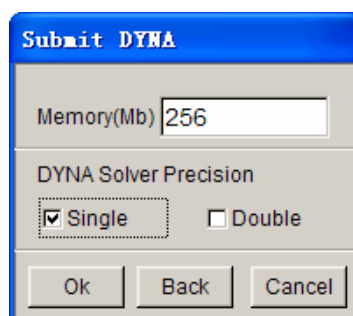


Figure 3.8.2 Select Memory dialog window

eta/DYNAFORM locates LS-DYNA by the given directory. Refer to Section 14.1, ANALYSIS PARAMETERS, for detailed information. If no solver or incorrect solver is assigned, the SELECT SOLVER window, as shown in Figure 3.8.3, will pop up after hitting Ok button from the Submit DYNA dialog window.

This function allows the user to assign the LS-DYNA solvers, as well as MSTEP solver.

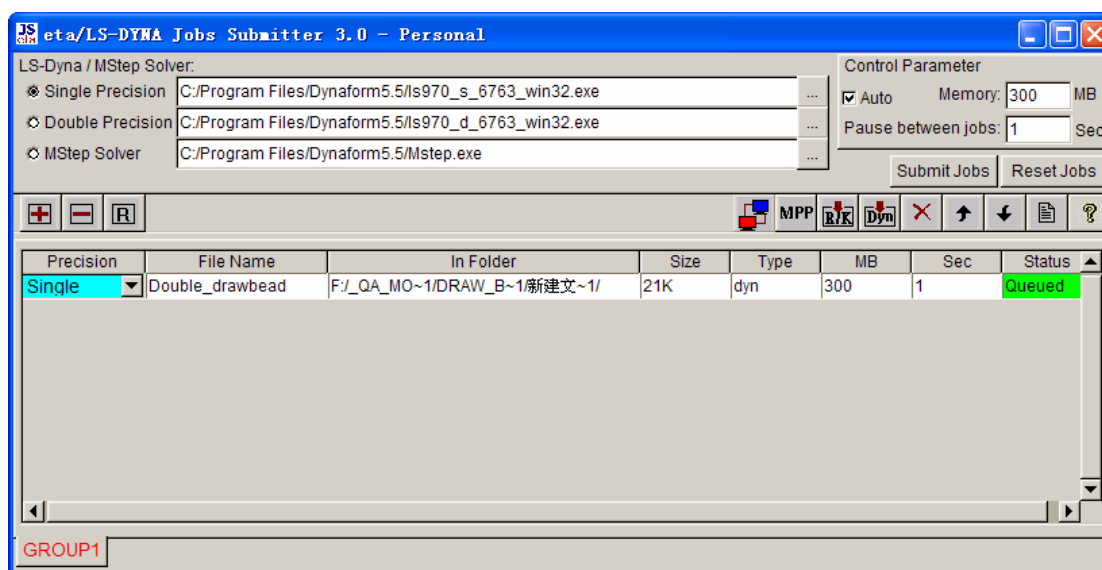


Figure 3.8.3 Select Solver window

### 3.9 RUN DYNA RESTART

This function enables the user to submit a LS-DYNA restart file to the solver.

1. eta/DYNAFORM prompts the user to select the restart file from the Select DYNA Restart file window as shown in Figure 3.9.1.

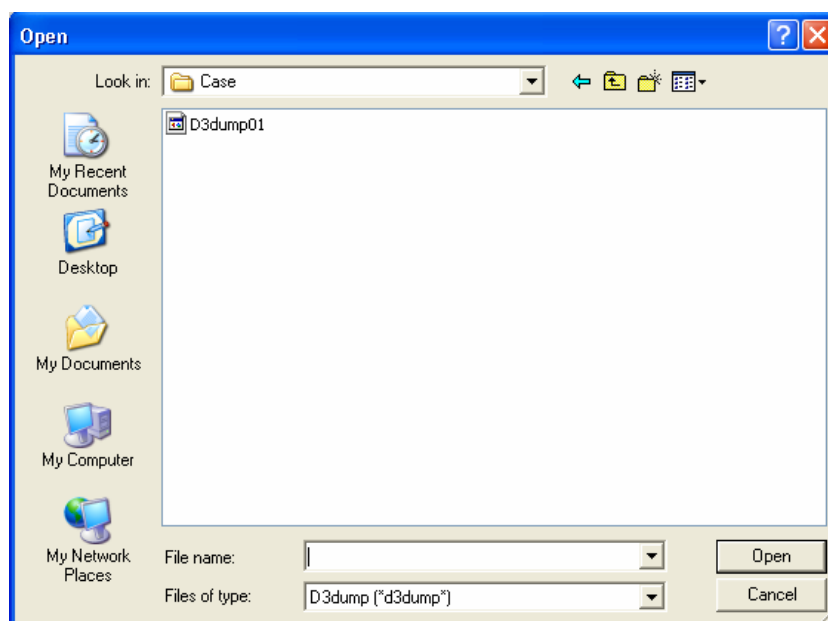


Figure 3.9.1 Select DYNA Restart File window

**NOTE:** Refer to the *LS-DYNA User's Manual* for more information about the DYNA restart file (*d3dump* and *runrsf*).

2. Double click the left mouse button on the desired file, or click OK to continue.
3. The SELECT MEMORY window appears, as shown in Figure 3.9.2.



Figure 3.9.2 Select Memory

**NOTE:** The memory allocation should be set equal to the memory allocation in the original run.

Refer to Section 3.8, SUBMIT DYNA FROM INPUT DECK, and Section 14.1, ANALYSIS PARAMETERS, for more information about the SELECT SOLVER.

## 3.10 PRINT SETUP

This function allows the user to define the default settings for printing postscript files. The options are shown in Figure 3.10.1.

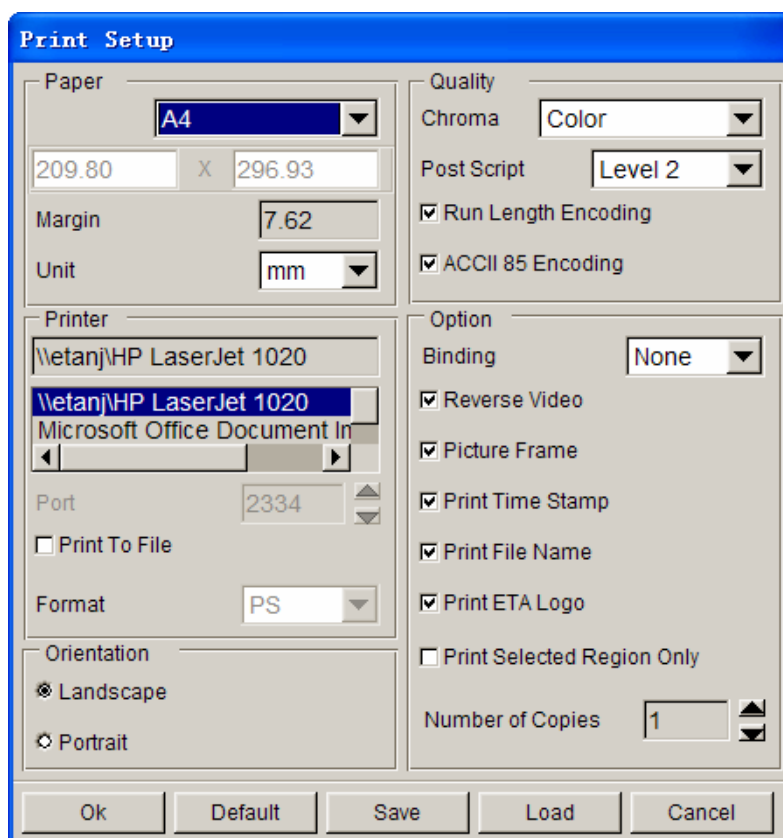


Figure 3.10.1 Print Setup window

### 3.10.1 PAPER

The user can specify paper and/or margin size using the following procedures:

- Click on the drop down button at the top left to choose a paper size.

**NOTE:** The actual sizes are: **LETTER** - 8.5x11 inches; **LEGAL** - 8.5x14 inches; **EXECUTIVE** - 7.25x10.5 inches; **A4** - 8.26x11.69; **SPECIAL A4** - 8.26x14; and **BS** - 7.17x10.13. Users can also specify a paper size for a specific printer by entering numbers in the two fields below.

- Enter a number in the field next to MARGIN to define the paper's top and bottom margins. The program automatically determines the left and right margins in order to

maintain the original aspect ratio of the picture. This feature can also be used to scale the picture.

- Click on the drop down button next to UNIT to choose a unit (inches or mm).

### 3.10.2 PRINTER

The user may select a printer or a format to print to a file using the following procedure.

- Enter the printer name in the field, or select from the list below.
- Set the port number in the field. A systems administrator usually sets the port number.
- If PRINT TO FILE is toggled ON, the user can choose a format by selecting the drop down choice next to FORMAT.

*NOTE: If PRINT TO FILE is selected, the user will be prompted to enter a filename when the PRINT function is clicked instead of the postscript file being sent directly to the printer.*

### 3.10.3 ORIENTATION

This function sets the page orientation as landscape or portrait.

### 3.10.4 QUALITY

#### COLOR TYPE

Click on the drop down button next to CHROMA to select the color type.

#### POSTSCRIPT

There are two levels; level 2 is the default setting, while level 1 should be selected for printing using an older model PS printer.

**RUN LENGTH ENCODE** and **ASCII 85 ENCODE** are used to reduce the postscript file size. Usually the reductions are significant. (For PS Level 1 these are automatically turned off.)

### 3.10.5 OPTION

These options are selected for defining printer output and layout on the paper.

#### **BINDING**

This function leaves binding space along the top or left margins of the print and selects non-binding space.

#### **REVERSE VIDEO**

This function reverses the black and white colors of the image. In some cases, this feature affects only the background of the image (default = on = white background).

#### **BOUNDING BOX**

This function draws a line frame around the picture's border.

#### **PRINT STAMP TIME**

This function prints the current time in the lower right corner of the picture.

#### **PRINT FILE NAME**

This function prints the file name in the lower left corner of the picture.

#### **PRINT ETA LOGO**

This function prints eta/DYNAFORM in the lower right corner of the picture.

#### **PRINT SELECTED REGION ONLY**

This function allows the user to define a graphics region by using a drag window. Only the graphics within this region will be written to the graphics file. If the function is off (default), the whole graphics region will be dumped to the file.

*NOTE: The user will be prompted to define the region (drag window) after PRINT is selected.*

### 3.10.6 NUMBER OF COPIES

This function allows the user to print multiple copies.

### 3.10.7 DEFAULT

This function loads the system defaults that are saved in a ".DynaformHardcopyDefault" file located in the eta/DYNAFORM installation directory.

### 3.10.8 SAVE

This function saves the user-modified hardcopy options to the ".DynaformHardcopyDefault" file located in the user's home directory which is the current user's "My Document" on PC and is the user "HOME" directory on UNIX or LINUX.

### 3.10.9 LOAD

This function loads the user-defined defaults that have been saved to the file, "DynaformHardcopy Default," located in the user's home directory or in the eta/DYNAFORM install directory.

*NOTE: The user must load the defaults for each new session. Otherwise the system defaults are used.*

## 3.11 PRINT

This function creates a postscript file of the display area and sends the file to the printer (default) or to a file. Prior to printing, the postscript driver must be initialized to accommodate the eta/DYNAFORM software. The user can define the printing defaults as described in Section 3.10.

## 3.12 EXIT (CTRL+X)

This function allows the user to exit the program. eta/DYNAFORM prompts the user to save the active database prior to closing the program

## Chapter 4

# PART CONTROL

A part in an eta/DYNAFORM database file consists a set of lines, surfaces, and/or elements. Each part has a unique Part Identification number (PID). The part name is a string consisting of up to eight characters. Currently, the user can create up to 1000 parts in a database. The functions in the PART CONTROL menu are shown in Figure 4.1. The user can organize lines, surfaces, and elements utilizing the functions in the PART menu.

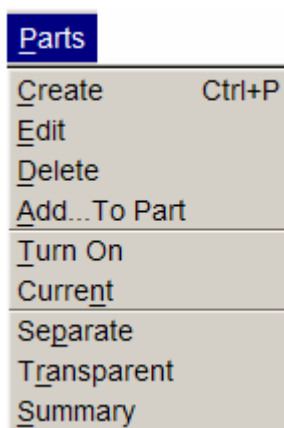


Figure 4.1 Part Control menu

A detailed description of each function is given in the following sections.

## 4.1 CREATE PART (CTRL+P)

This function enables the user to create a new part.

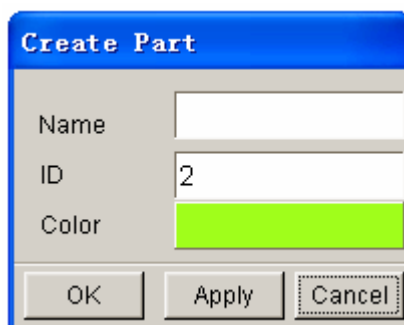


Figure 4.1.1 Create Part dialog box

As shown in Figure 4.1.1, the Create Part dialog box enables user to create a new part by the following procedures:

1. Type a new part name in the NAME field.
2. Type an ID number that has not been used by another part or use the default ID field.
3. Select the COLOR button to display the SELECT COLOR window allowing the user to choose a color. The color options are chosen from the Select Color dialog box as shown in Figure 4.1.2.

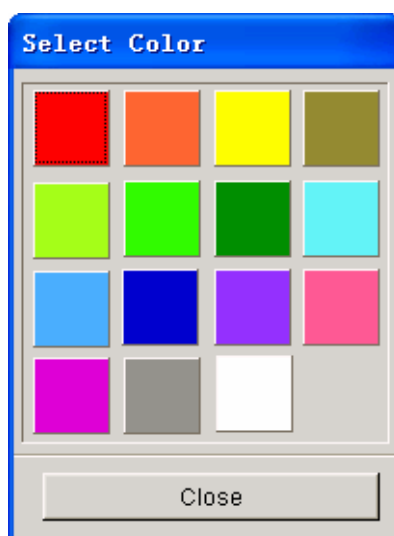


Figure 4.1.2 Select Color

4. Choose either the OK or APPLY button. Clicking on the OK button to create a new



part and exit the CREATE PART function. Clicking on the APPLY button to create a new part and remain in the CREATE PART function to continue creating new parts. The new part created is the current part and is shown in the DISPLAY OPTIONS window.

## 4.2 EDIT PART

The EDIT PART function is used to modify and delete parts as well as editing a part name, part ID, and color. The EDIT PART dialog box is illustrated in Figure 4.2.1.



Figure 4.2.1 Edit Part dialog box

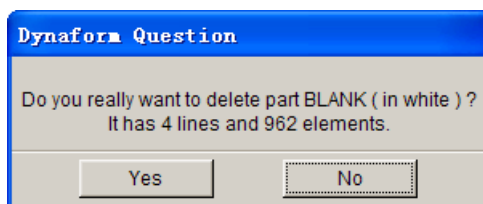
### 4.2.1 MODIFY

The MODIFY button enables user to edit the part name, ID, and/or color. The procedure is:

1. Use mouse cursor to select a part from the list.
2. Type in new part name and/or part ID
3. Click on MODIFY button to finish the modification.

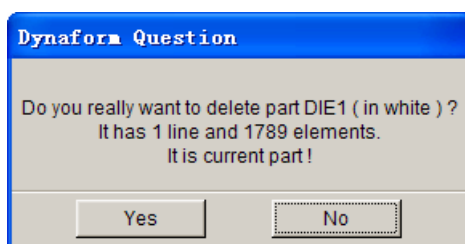
### 4.2.2 DELETE

The DELETE button enables user to remove a selected part from the list and database. A eta/DYNAFORM QUESTION window will appear to prompt the user to confirm or cancel the operation. If the part is not the current part, the system will pop up a question dialog box as shown in Figure 4.2.2. The user can confirm or cancel the removal of part.



Figures 4.2.2 Delete the non-current part question dialog box

If the part is current part, the system will pop up a question dialog box as shown in Figure 4.2.3.



Figures 4.2.3 Delete the current part question dialog box

### 4.3 DELETE PART

The function enable user to remove one or more than one selected part from the database.

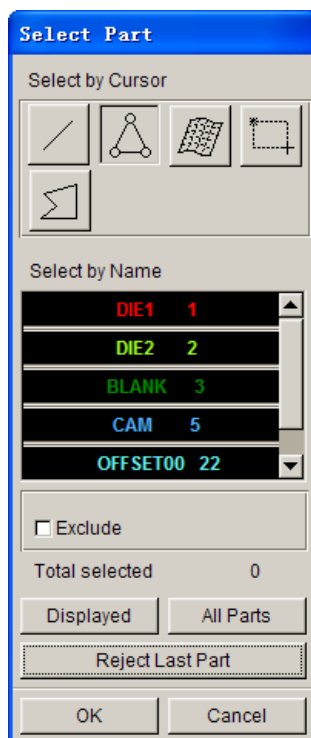


Figure 4.3.1 Delete Part(s) dialog box

As shown in Figure 4.3.1, user can select part(s) from the part list of Select Part dialog box. Different options such as through the Line, Element, Surface, Drag Window and Multi-Point Region are also available for part selection.

If user wants to delete all parts in the database, he/she can click on the ALL PARTS button to delete all parts in current database. Upon activating the function, a eta/DYNAFORM question dialog box as Figure 4.3.2 shown will appear. Click on Yes button to remove all parts or No to cancel the operation.

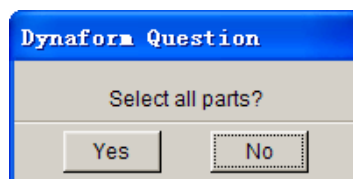


Figure 4.3.2 eta/DYNAFORM Question dialog box

## 4.4 ADD...TO PART

This function allows the user to organize lines, surfaces, and elements from one part to another. The dialog box is illustrated in Figure 4.4.1.

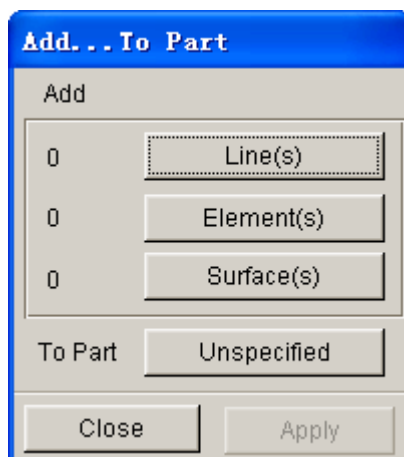


Figure 4.4.1 Add... To Part dialog box

### 4.4.1 ADD LINE

The ADD LINE function is used to organize lines. The operation procedure is listed as following:

- Click on the LINE(S) button to open SELECT LINE dialog box as shown in Figure 4.4.2.

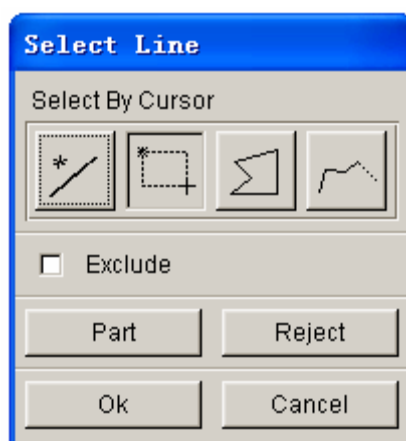






Figure 4.4.2 Select Line dialog box

- The SELECT LINE dialog box offers four options for line selection. Once selected, the line will be highlighted in white.

1. Select  icon (Pick Line) to pick line using mouse cursor.
2. Select  icon (Drag Window) to pick lines inside a rectangular window. The user defines the rectangle by selecting two diagonal points.
3. Select  icon (Multi-point Region) to pick lines inside a closed polygon. Each left mouse button click defines a vertex and a middle button click closes the polygon.
4. Select  icon (Chain Line) to pick lines using mouse cursor. All lines that are chained (connected) to the selected line will be selected.
5. Click on PART button to pick all lines in the selected part.

- To Deselect lines,
  1. Toggle on the EXCLUDE checkbox. The user can exclude the selected lines using the line selection method mentioned above.
  2. Select REJECT, and the last line selection will be rejected.
- Select OK to complete the operation or CANCEL to abort the operation.
- To select a target part, click on Unspecified button from Add...to Part dialog box,

following by selecting a part from the part list.

- Then, click on APPLY button to complete moving lines from one part to a designated part.
- To abort the operation, click on CLOSE button.

#### 4.4.2 ADD ELEMENT

- Click on ELEMENT(S) button to open the SELECT ELEMENTS dialog box as shown in Figure 4.4.3. There are three options available for selecting elements in the display screen and/or active database. They are SELECT BY CURSOR, SELECT BY, selection by Displayed (elements on the display screen) and All Elements (in active database).

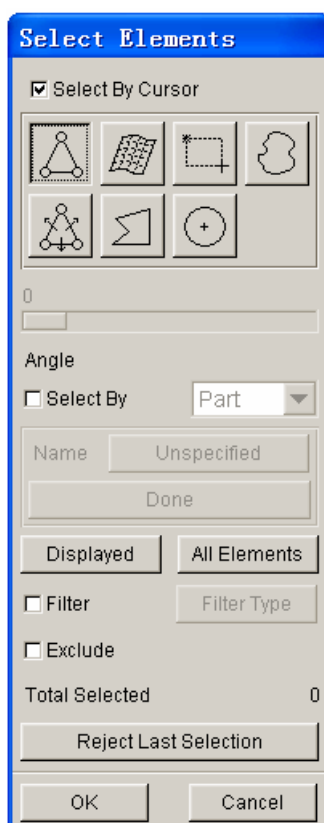








Figure 4.4.3 Select Element dialog box

- If the checkbox for SELECT BY CURSOR is toggled on, the user can select elements using the following methods.



1. Click on (Element) icon to select an element using mouse cursor and/or

pressing the left mouse button and move the mouse cursor over the desired elements to select multiple elements.

2. Click on  (Surface) icon to pick elements on a surface.
3. Click on  (Drag Window) icon to pick elements inside a window defined by two diagonal corners.
4. Click on  (Free Region) icon to pick elements inside a free hand region.
5. Click on  (Spread) icon to pick elements by chained approach based on spread angle which is the angle difference between normal vector of an element and normal vector of its adjacent elements. The slider can be moved to adjust the spread angle. If the angle formed by the normal vectors of the selected element and the adjacent element is less than the selected angle, the adjacent element is selected.
6. Click on  (Multi-Point Region) icon to pick elements inside a polygon. Click the left mouse button in a sequence to define the polygon. Click the right mouse button to reject the last point. Click the middle mouse button to complete polygon definition and element selection.
7. Click on  (Circular Region) icon to pick elements inside a circle. Click the left mouse button to define the center of the circle and hold the left mouse button. Move the mouse to define the circle. Release the left mouse button to complete definition of circle and element selection.

### SELECT BY

- If the SELECT BY option is toggled on, two approaches can be used to select elements. The user may select elements by Part and/or Range by clicking on the drop down button as illustrated in Figure 4.4.4.



Figure 4.4.4 Select element by Part and/or Range

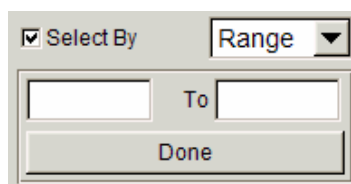


Figure 4.4.5 Select element by Range

#### 1. SELECT BY PART

Refer to Section 4.4 for procedure to select a part. All elements in the part will be selected when this option is chosen. Then, click on the Done button to complete the operation. Figure 4.4.4 illustrates select element using Part option.

#### 2. SELECT BY RANGE

Type the range of element ID in the input box as illustrated in Figure 4.4.5, following by clicking on Done button to complete the operation. All elements within the desired range will be selected.

### FILTER

- If the Filter option is toggled on, user can select the specifically element type through this option. User can click the FILTER TYPE button to set the element type. The Element Filter Type dialog box is illustrated in Figure 4.4.6.

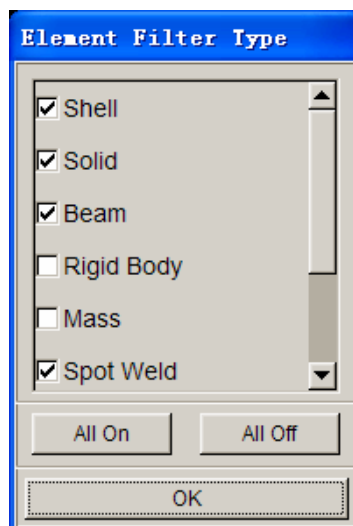


Figure Element Filter Type dialog box

There are nine types of element in the Element Filter Type for user to filter. They are Shell, Solid, Beam, Rigid Body, Mass, Spot Weld, Spring/Damper, Thick Shell and Plotel. User can toggle off the unneeded element types to reduce the range of selection.

- User can select all elements in the display screen by clicking on the DISPLAYED button.

- The total number of selected elements will be shown in the window. The EXCLUDE option is used to remove selected elements, while the REJECT function is used to deny last selected elements.
- Select OK to complete the element selection and return to the Add...To PART dialog box.

### 4.4.3 ADD SURFACE

- Click on the SURFACE(S) button to open SELECT SURFACE dialog box as illustrated in Figure 4.4.7.

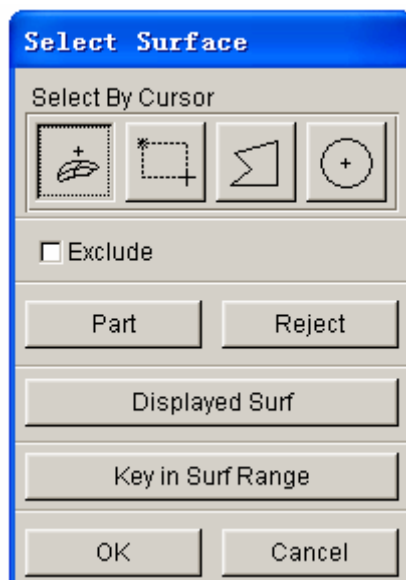


Figure 4.4.7 Select Surface dialog box

- **DISPLAYED SURF**  
This function is used to select all displayed surfaces in the display area.
- **KEY IN SURF RANGE**  
This function enables surface selection by entering the range of surface numbers. As illustrated in Figure 4.4.8, the user can enter surface numbers and increment.



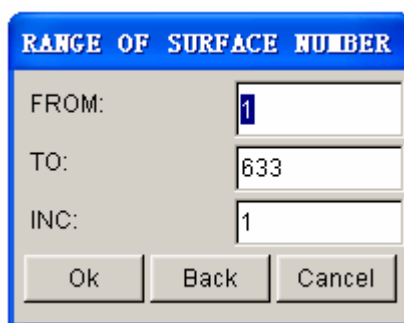


Figure 4.4.8 Define Surface Range dialog box

Other options for surface selection resembles those described in Section 4.4.1.

#### 4.4.4 SELECT THE TARGET PART

After selecting lines, surfaces, and/or elements, the user needs to assign the target parts which will contain the selected entities. Then, click on the APPLY button to complete the operation. Else, the user can select CANCEL to abort the operation.

### 4.5 TURN ON

This function enables the user to show/hide the selected parts. The PART TURN ON/OFF dialog box, as shown in Figure 4.5.1, appears once the function is selected.

- The user can show/hide the selected part using any of the 3 options provided in Turn Part On/Off dialog box: selecting a line, element, or surface. In addition, the part can also be shown/hidden by using mouse cursor to pick on the parts listed in the SELECT BY NAME list.

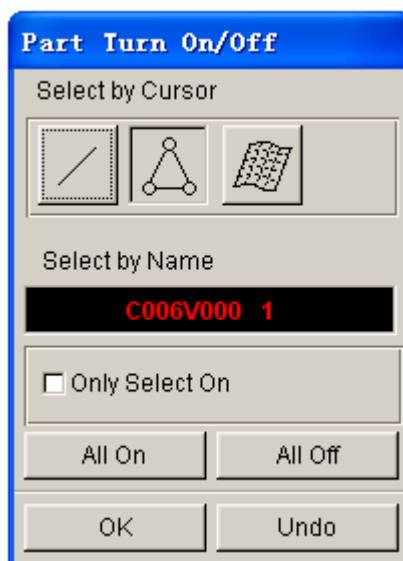


Figure 4.5.1 Turn Part On/Off dialog box

- Only the selected part will be shown in the display area if the **ONLY SELECT ON** option is toggled on.
- **ALL ON**  
All parts will be shown in the display area.
- **ALL OFF**  
All parts will be hidden in the display area.

## 4.6 CURRENT PART

This function allows the user to define the current part. All lines, surfaces, and elements created by users are automatically stored in the current part. The name of current part is in the **DISPLAY OPTIONS** window. As illustrated in Figure 4.6.1, the user can select the current part from the Current Part dialog box.

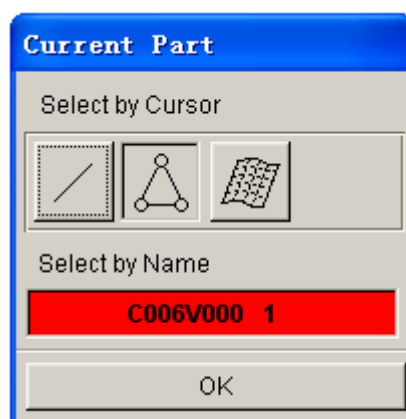


Figure 4.6.1 Current Part dialog box

- The user can select current part using any of the 3 options provided in Current Part dialog box: selecting a line, element, or surface. In addition, the part can set as active by using mouse cursor to pick on the part listed in the SELECT BY NAME list.

## 4.7 SEPARATE PART

This function allows the user to separate two parts that share common nodes and nodal IDs along the joint line. After separation, each part carries its own nodes and nodal ID along the joint line.

The functions in the SELECT PARTS dialog box resemble to those described in Section 4.2. Select the ALL PARTS button for separating all parts or selected parts. Click OK to finish separating parts.

## 4.8 TRANSPARENT

This function enables the user to set transparent parts during the shading operation. The user can also adjust the degree of transparency for each part.

## 4.9 SUMMARY

This function enables the user to list statistic data regarding the geometry, material, and interface information of the selected parts. An example of a part summary table is shown in Figure 4.9.1.

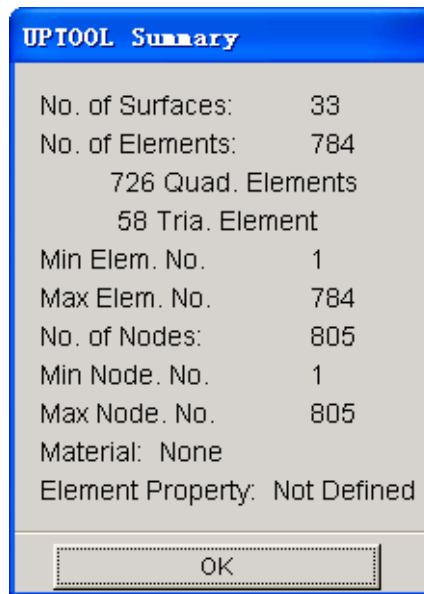


Figure 4.9.1 Part Summary dialog box

## Chapter 5

# PREPROCESS

The functions provided in this menu allow the user to build and modify a model, generate a model with elements, and also check and add the boundary conditions to a model. The submenus are shown in Figure 5.1.

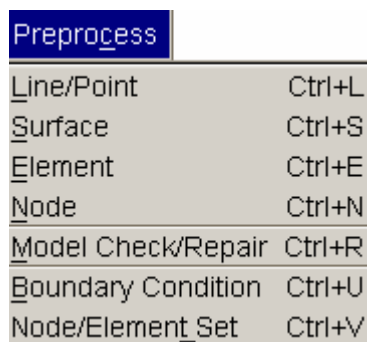


Figure 5.1 Preprocess Menu

A detailed description of these functions is given in the following sections.

Many functions of dialog and submenu windows are similar to those described in previous chapters. References of the window locations are discussed in the appropriate sections.

### 5.1 LINE/POINT (CTRL+L)

The functions in this dialog window are used to build line data. The user can move the cursor to an icon to see the function name. The functions in Line/Point menu are shown in Figure 5.1.1.

A detailed description of these functions is given in the following sections.





At the intersection of two lines.



On the workplane. Click on the workplane icon to display the grids on the display area and activate Workplane and Grid Setting button. The user can generate a line on the workplane by clicking the left mouse button on two or multiple points.

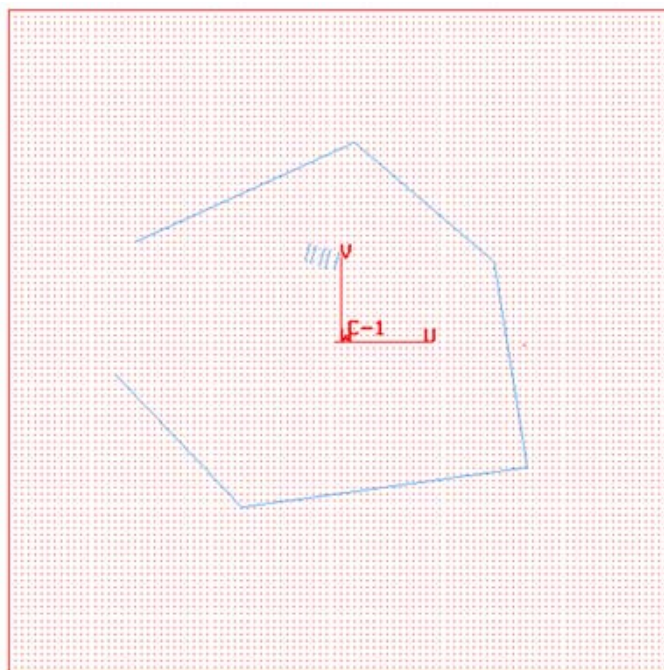


Figure 5.1.2 Working Plane and Grid

### WORKING PLANE AND GRID SETTING

The user can define the setup of the workplane and grid from the Grid Setting dialog box as illustrated in Figure 5.1.3.

#### PLANE COORDINATE:

Set the direction of the workplane, the default workplane is x-y plane of the Globe coordinate system. The user can select the coordinate plane of the view direction or LCS from the list as the current workplane.

#### LIMIT BOX:

Set the size of the grids displayed on the workplane. The user can define the maximum and minimum coordinate of the display grids.

## DIVISION

Set the width of the grid division.

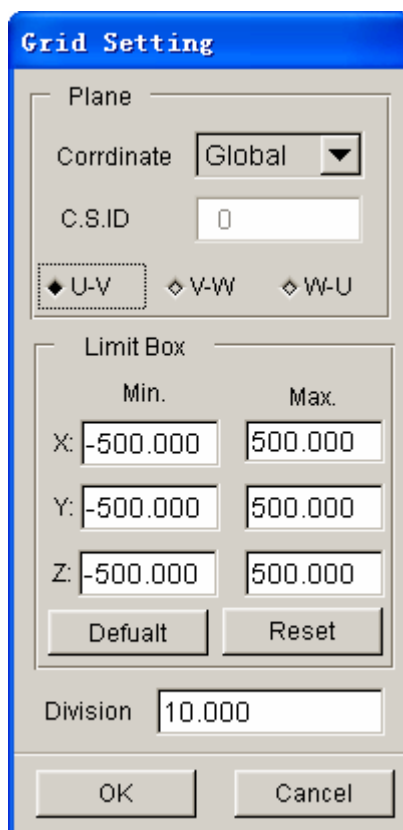


Figure5.1.3 Workplane and Grid Setting dialog box

1. To create a point by its coordinates.
  - Select the LCS or ABS option for the coordinates in the current local coordinate system or global system. Refer to Section 12.14, LOCAL COORDINATE SYSTEM.
  - Select the XYZ or DXYZ option.
    - If XYZ is selected, the input values of U, V and W are the coordinates.
    - If DXYZ is selected, the input values of DU, DV and DW are increments of the previously created point.
  - Type in each value in the corresponding field.
  - Select APPLY INPUT VALUE to confirm the values and create the point.



2. To cancel the last created point
  - Click REJECT once.
3. To finish the definition of the line
  - Select OK or click on the middle mouse button.
4. To close the dialog box
  - Select CANCEL or click on the right mouse button. The selected points will be ignored when the CANCEL button is selected.

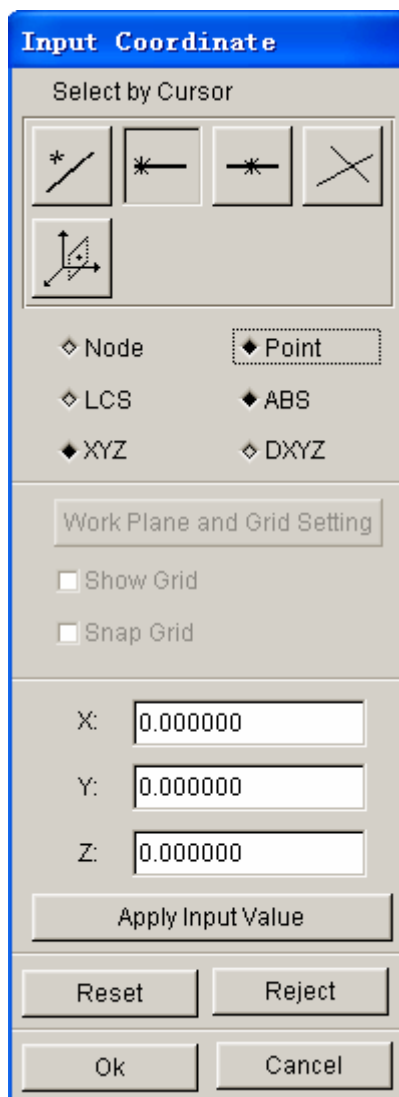


Figure 5.1.4 Input Coordinate dialog box

## 5.1.2 CREATE ARC



This function allows the user to generate arcs. As illustrated in Figure 5.1.5, arcs can be generated using three approaches.

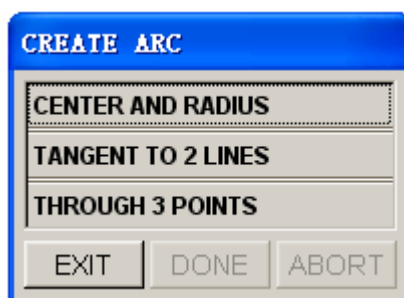


Figure 5.1.5 Create Arc dialog box

## 1. CENTER AND RADIUS

- Define a LCS. Refer to Section 12.14, LOCAL COORDINATE SYSTEM, for information about defining LCS. The center of the arc will be positioned at the origin of LCS, and the arc will be in the UV plane of the LCS.
- Key in the values in the ARC PARAMETER dialog box as illustrated in Figure 5.1.6.

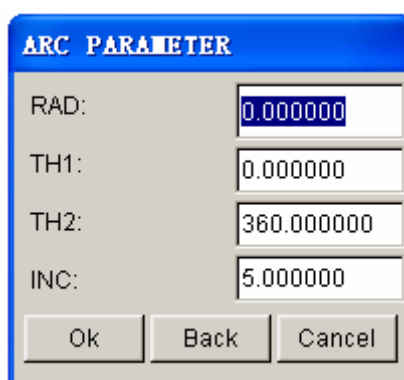


Figure 5.1.6 Arc Parameters dialog box

**RAD:** Radius of arc

**TH1:** Beginning angle from U-axis

**TH2:** Ending angle from U-axis

**INC:** Angle increment between points (default is 5 degrees) on an arc

## 2. TANGENT TO 2 LINES

This function allows the user to create an arc that is tangent to 2 lines.

- Select two intersecting lines. Refer to Section 4.4.1, PART/ADD LINE, for more information about selecting lines.
- Enter the radius in the input field of RADIUS OF ARC dialog box as illustrated in Figure 5.1.7.

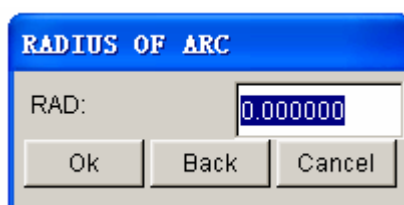


Figure 5.1.7 Radius of Arc dialog box

## 3. Create by THROUGH 3 PTS

This function allows the user to create an arc through three non-collinear points/nodes.

- Select three points/nodes. After the third point/node is selected, the arc and its center will be generated.

### 5.1.3 CREATE SPLINE



This function allows the user to generate a spline curve through multiple points and/or nodes. A minimum of three points or nodes is required.

The user selects the points and/or nodes using the INPUT COORDINATE function. Click on Ok button to complete spline curve generation.

#### 5.1.4 DELETE LINE



This function allows the user to remove lines in the database.

- Use mouse cursor to select line. Refer to Section 4.4.1, PART/ADD LINE, for more information about selecting lines.

#### 5.1.5 COPY OR TRANSFORM LINE



This function allows the user to generate a set of duplicate lines. The user can simultaneously translate and rotate the duplicated lines. The COPY LINES dialog box is illustrated in Figure 5.1.8.

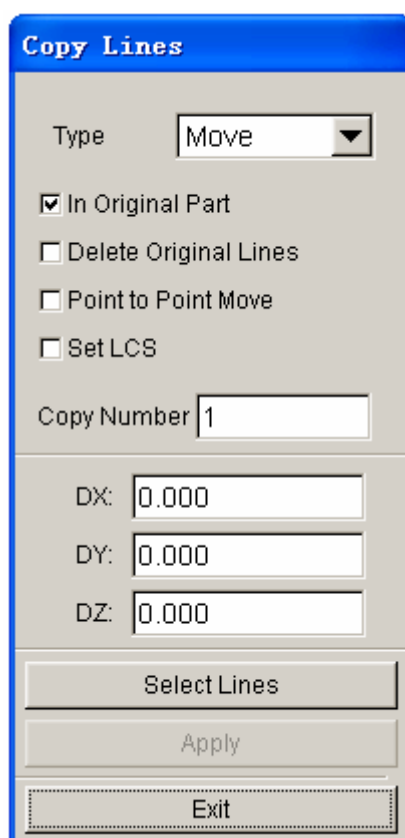


Figure 5.1.8 Copy/Transform Line dialog box

1. Select lines. Refer to Section 4.4.1, PART/ADD LINE, for more information about

- selecting lines.
2. Choose a type of transformation, either MOVE or ROTATE.
  3. Key in the number of copies.
  4. Toggle on checkbox for IN ORIGINAL PART. If IN ORIGINAL PART is not checked, the transformed lines will be stored in the current part.
  5. Toggle on checkbox for DELETE ORIGINAL LINES to erase the original line(s) after transformation.
  6. Toggle on/off POINT TO POINT MOVE option. The POINT TO POINT MOVE option will be described later. This option is only active for the translation.
  7. Toggle on/off SET LCS option. The SET LCS will be described.
  8. Type in the increments DX, DY, DZ for the translation or type in the angle increment DX for rotation about the W-axis.
  9. Click on APPLY button to complete the operation.
- If POINT TO POINT MOVE option is selected, the INPUT COORDINATE dialog box will be displayed and the user has to select two points. The translation will be based on the differences of three coordinates of selected points.
  - If SET LCS option is selected, the transformation for the selected line(s) will be done in the defined local coordinate system. After the user selected line(s) and clicked the APPLY button, the program will pop up the LCS dialog box for user to define a local coordinate system. For more detailed information about how to create a LOCAL COORDINATE SYSTEM, please refer to Section 11.14.
  - If SET LCS option is not selected, the LCS dialog box will not be shown. The transformation will be done in the global coordinate system.

10. Select OK to dismiss the COPY LINE dialog box.

### 5.1.6 MODIFY LINE



This function allows the user to modify a line by moving its points.

- Select a line from the display area. Refer to Figure 4.4.2 of Section 4.4.1, PART/ADD, for more information.
- Select a point on the line to modify.
- Input the new location for the selected point in the INPUT COORDINATE dialog box.
- Select Ok button in the INPUT COORDINATE dialog box to complete the operation.

### 5.1.7 ADD POINT



This function allows the user to add points to an existing line.

There are two ways to add a point:

#### 1. BETWEEN 2 POINTS

This option creates a specified number of new points between two selected endpoints.

- Select two points by cursor on a line.
- Enter the desired number of points between two endpoints in the next dialog box.

#### 2. CURSOR LOCATION

The user may create new points anywhere on a line by using the options in the INPUT COORDINATE dialog box.

### 5.1.8 COMBINE LINE



This function allows the user to stitch multiple lines into a single line.

- Click on the Combine Line icon from the LINE/POINT dialog box. The program will pop-up a dialog box, as illustrated in Figure 5.1.9, for the user to select the gap value between lines. If the user doesn't toggle on the line gap option, all select lines will be combined.

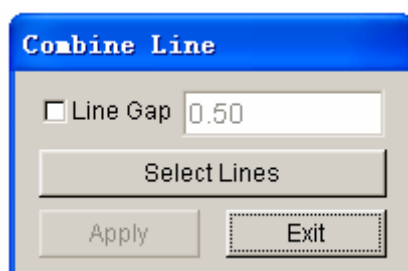


Figure 5.1.9 Combine Line

- Line Gap option enables the user to combine all the selected lines which have a gap value that is less than the specified line gap.
- Select Lines button enables the user to select lines.
- Click on the Apply button complete the operation.
- The user can click Select Lines to continue the operation or click on EXIT button to exit Combine Line function.

### 5.1.9 SPLIT LINE



The user can split a line into two lines. The Select Line dialog box is used to select a line from the display area, while the INPUT COORDINATE dialog box is used to determine split location.

1. Select a line using one of the functions in the SELECT LINE dialog box. Refer to Section 4.4.1, PART/ADD LINE, for more information.
2. Select the split location using one of the functions in INPUT COORDINATE dialog box. Refer to CREATE LINE, Section 5.1.1, for more information.

### 5.1.10 EXTEND LINE



This function allows the user to extend a line by a specified length from the endpoint of a line.

- Select a line from the display area using mouse cursor.
- Select one endpoint of the selected line.
- Enter a real number as the extended length in the Length of Extend dialog box as illustrated in Figure 5.1.10.

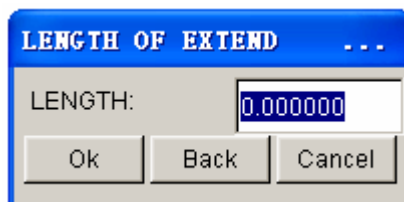


Figure 5.1.10 Length of Extend dialog box

### 5.1.11 MIRROR LINE



This function allows the user to mirror a set of lines with respect to a given plane. The MIRROR LINES dialog box is illustrated in Figure 5.1.11.



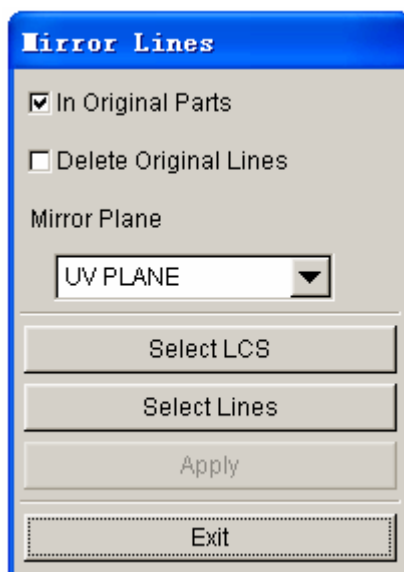


Figure 5.1.11 Mirror Line dialog box

- Select line(s). Refer to Section 4.4.1, PART/ADD LINE, for more information.
- Toggle on/off the option checkbox.

#### **IN ORIGINAL PARTS**

The created lines are stored in the original part if the checkbox is toggled on. If the checkbox is toggled off, the created lines are stored in the current part.

#### **DELETE ORIGINAL LINES**

The original lines are deleted if the checkbox is toggled on.

- Select LCS. Refer to Section 11.14, COORDINATE SYSTEM, for more information.
- Select a coordinate plane as the mirror plane.
- Click on the APPLY button to complete the operation
- Click on the Exit button to dismiss the Mirror Lines dialog box.

### 5.1.12 OFFSET LINE



This function is used to create an offset line from an existing line. Usually, it is used to offset a planar line. The offset line will be created in the same plane.

- Select a line from the display area.
- Define a coordinate system to determine the U-V plane.
- Enter a positive offset distance in input field of the Offset Distance dialog box as illustrated in Figure 5.1.12.
- The eta/DYNAFORM Question dialog box as illustrated in Figure 5.1.13, is displayed. There is an arrow in the display area indicating the offset direction.

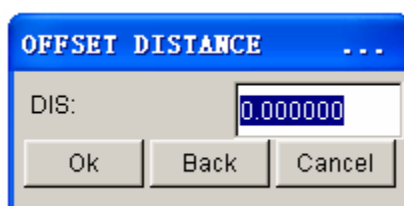


Figure 5.1.12 Offset Distance dialog box

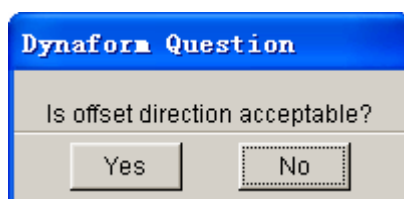


Figure 5.1.13 eta/DYNAFORM Question dialog box

Click on YES button to accept the direction.

Click on NO button to reverse the direction.

**5.1.13 SCALE LINE**

This function allows the user to enlarge or reduce the size of lines.

- Select a line. Refer to Section 4.4.1, PART/ADD LINE, for more information.
- Define a coordinate system for scaling. Refer to Section 12.14, COORDINATE SYSTEM, for more information.
- Enter the desired scale factors in the input fields in the SCALE FACTOR dialog box as illustrated in Figure 5.1.14.

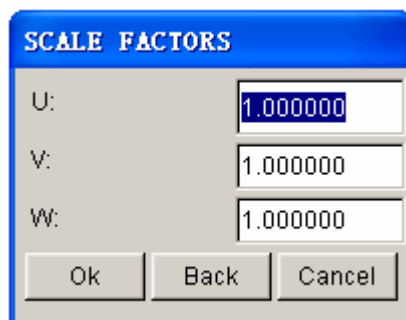


Figure 5.1.14 Scale Factor dialog box

**5.1.14 SHOW LINE**

The user may identify any existing line and its line direction using this function. The selected line is highlighted and the starting point is labeled with a circle. The subsequent points are represented with arrowheads and the number of points on the selected line is printed in the message prompt window.

As illustrated in Figure 5.1.15, two approaches can be used to show a line.

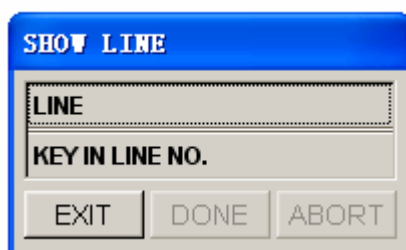


Figure 5.1.15 Show Line dialog box

1. LINE  
Select line using mouse cursor.
2. KEY IN LINE NO.  
Enter the line number following by clicking on the OK button.

#### 5.1.15 REVERSE LINE DIRECTION



Each line has a direction defined from the starting point to the ending point of the line. This function allows the user to reverse the direction of line.

After the function is selected, the user can select lines using mouse cursor. Once the lines are selected, the direction of the selected lines is reversed automatically.

#### 5.1.16 RESPACE LINE



This function allows the user to redistribute the points on the selected line. The dialog box is illustrated in Figure 5.1.16.

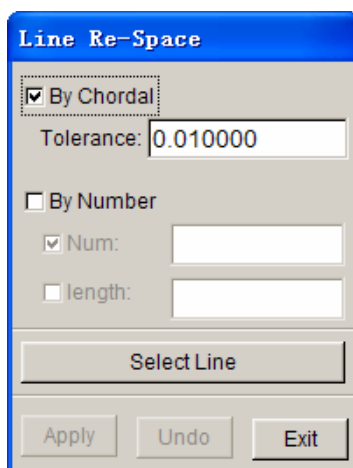


Figure 5.1.16 Re-space Line dialog box

- Select the line using mouse cursor. The Line Re-space dialog box will pop-up as illustrated in Figure 5.1.16. There are two options available for user to decide the density of point on line.

#### BY CHRODAL

This option enables user to adjust the number of points on line by inputting the tolerance. The default tolerance is 0001.

#### BY NUMBER

This option enables user to adjust the number of points on line by inputting the point number or the length of between two adjacent points on line.

### 5.1.17 PROJECT LINE



This function allows the user to project lines or points onto a selected surface.

- Select a surface. Refer to Section 4.4.3, PART/ADD SRUFACE, for more information.
- The user must define a coordinate system. The projected direction is along the W-axis of the defined coordinate system.

- PROJECT LINE/POINT dialog box as illustrated in Figure 5.1.17 is displayed.

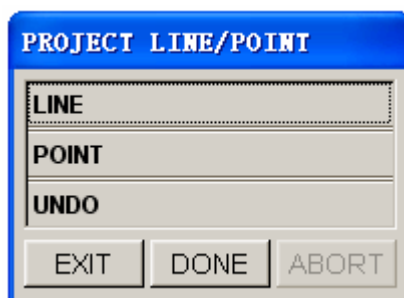


Figure 5.1.17 Project Line

- The selected lines or points will be projected on the selected line along the given W-axis. If there is no projection point found for some lines or points, an error message will be printed in the message prompt area:

***NO PROJECTION FOUND FOR XX POINTS***

XX indicates the number of points that have no projection point.

#### 5.1.18 SECTION THROUGH LINE



This function generates a line consisting of the intersection points of a user-defined plane and a selected set of lines.

- Define a local coordinate system. The section plane is parallel to the UV-plane.
- Enter a real number that determines the position of the section plane from the UV-plane.
- Select lines using mouse cursor. The section line will be generated in the current part.

#### 5.1.19 F E. BOUNDARY LINE



This function allows the user to generate a line around the boundary of a finite element mesh. The dialog box is illustrated in Figure 5.1.18.

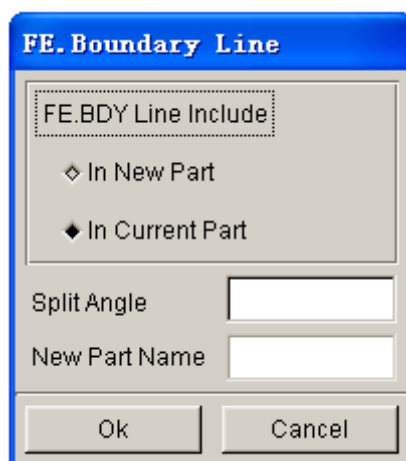


Figure 5.1.18 FE.Boundary Line dialog box

### **IN NEW PART**

This option includes the boundary line in a new part. The new part name is entered in the input field of NEW PART NAME. If IN NEW PART is toggled off, the NEW PART NAME input field is disabled.

### **IN CURRENT PART**

This includes the boundary line in the current part (default).

### **SPLIT ANGLE**

The generated line will be split at the corners of which the angles are less than the split angle. The default value is zero.

### **5.1.20 BRIDGE LINE**



This function generates a line connecting two selected lines. The program generates a b-spline curve that is continuous at the end points of the selected lines. The generated line is included in the current part. The program displays a select line dialog box as illustrated in Figure 5.1.19 and prompts the user to select two lines.

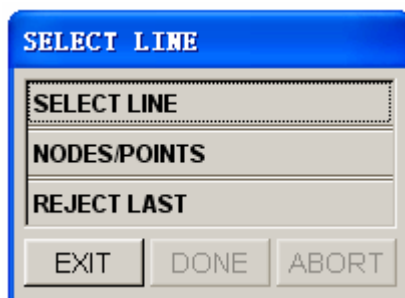


Figure 5.1.19 Select Line dialog box

There are three options available:

#### **SELECT LINE**

This is the default option. The user may start selecting lines from the screen without selecting this function. The program will use the end point of the selected line near the cursor to connect the bridge line.

#### **NODES/POINTS**

This option allows the user to define the line from selecting nodes or points on the screen. The program will display an INPUT COORDINATES dialog box to prompt the user to select nodes or points to define the line.

#### **REJECT LAST**

This option is used to reject the last selected line or defined line.

Figure 5.1.20 illustrates the original lines and the generated bridge line.



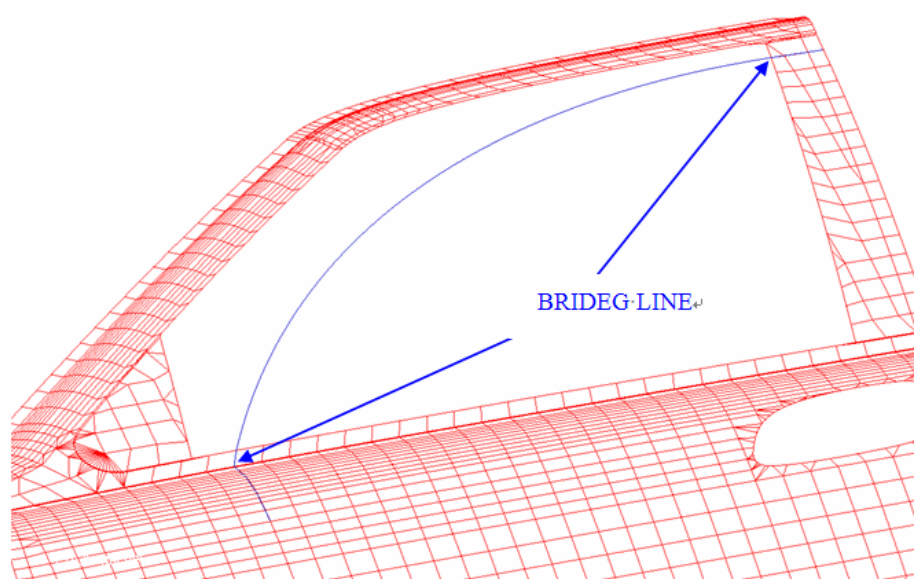


Figure 5.1.20 Bridge Line

## 5.2 SURFACE (CTRL+S)

The functions in this menu allow the user to create and modify surfaces in eta/DYNAFORM. The dialog box is illustrated in Figure 5.2.1.

eta/DYNAFORM uses two sets of dotted lines to show the contour of a surface. These lines are called U-V lines.

- **U-V LINE**

This function controls the on/off display of the U-V line on the surface.

- **SURFACE NORMAL**

This function controls the on/off display of the normal direction of the surface while the U-V line function is on.

A detailed description of these options is given in the following sections.



- LINE allows the user to select lines (default).
- LINE SEGMENT – This option allows the user to select several line segments to form a line. After the desired line segments are selected, click on the Ok button in Select Line dialog box.
- POINTS/NODES – This option allows the user to select two points/nodes to form a line. Select the lines, the line segments, or two points/nodes from which the surface will be created. After the second line is selected or formed from line segments or two points/nodes, the surface will be generated.

### 5.2.2 CREATE 3L



This function allows the user to create a surface from three selected lines. The procedure is similar to those described in Section 5.2.1.

### 5.2.3 CREATE 4L



This function allows the user to create a surface from four lines. The procedure is similar to those described in Section 5.2.1.

**NOTE:** *Select a line in a clockwise or counterclockwise direction to define the surface normal direction. The direction of the surface normal dictates the direction of the plate element normals generated from the surface.*

### 5.2.4 REVOLUTION SURFACE



This function allows the user to create a surface by rotating a selected line as a section (generatrix) about a vector (axis of revolution).

- Select the endpoints of the axis in the INPUT COORDINATE dialog box. A vector

will be displayed after the second point is selected.

- Select a line as the section (generatrix).
- Enter the start and end angles in the dialog box as illustrated in Figure 5.2.3. Select OK, and the surface will be generated after the dialog box is closed.

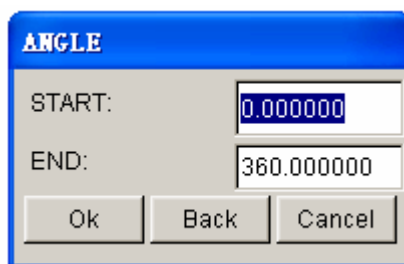


Figure 5.2.3 Revolution Angle dialog box

## 5.2.5 SWEEP SURFACE



This function allows the user to create a surface by sweeping a pattern (a selected line) along another direction line. The sweep direction follows the direction of the direction line.

- There are two types of SWEEP operations:
  1. NORMAL SWEEP  
The section line rotates along the normal direction of the direction line while sweeping.
  2. RIGID SWEEP  
The section line does not rotate while sweeping.
- The procedure of selecting lines is similar to those described in Section 5.2.1.

### 5.2.6 SHOW SURFACE



This function allows the user to highlight the selected surface and its UV lines. The dialog box is illustrated in Figure 5.2.4.

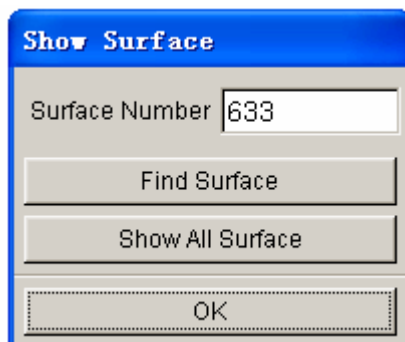


Figure 5.2.4 Show Surface dialog box

There are several ways to show surface:

1. Enter the surface number in the field; then select FIND SURFACE. The surface ID number will be shown.
2. Select SHOW ALL SURFACES. The surface ID of the displayed surfaces will be shown.
3. Select the surface in the graphics display window directly (default).

### 5.2.7 DELETE SURFACE



This function allows the user to remove the selected surface from the database. Refer to Section 4.4.3, PART/ADD SURFACE, for information about selecting surfaces.

### 5.2.8 TRANSFORM SURFACE



This function allows the user to move and/or rotate the selected surfaces.

- Define a new coordinate system by using the options in the LCS dialog box.
- Select MOVE or ROTATE.

If MOVE is selected, the user can enter increments of U, V and W in the UVW Increments dialog box.

If ROTATE is selected, the user can enter increments of the angle in the Angle Increment dialog box. The surface will be rotated about the W-axis.

- Select the desired surface by using the options in the SELECT SURFACE dialog box.
- The user can select AGAIN to repeat the operation, REVERSE OPERATION to reverse the operation, or SELECT SURFACE to select another surface.

### 5.2.9 COPY SURFACE



This function allows the user to copy the selected surface(s).

- Select a surface by using options in the SELECT SURFACE dialog box.
- Define a new coordinate system by using options in the LCS dialog box.
- Enter the number of copies along the copy direction.
- Select either MOVE and/or ROTATE.

- Enter the increments of U, V, W or angle.
- Select YES or NO in the eta/DYNAFORM Question dialog box, as illustrated in Figure 5.2.5 to determine the part to store the surfaces.

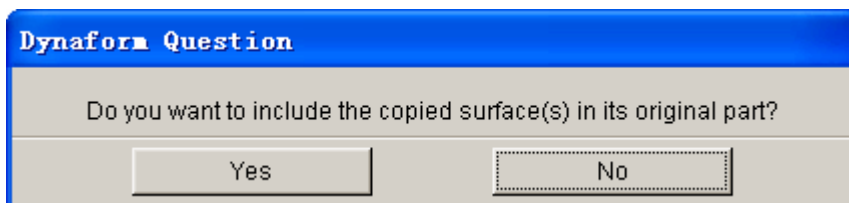


Figure 5.2.5 DYNAFORM Question dialog box

### 5.2.10 MIRROR SURFACE



This function allows the user to mirror selected surfaces with respect to a given plane.

- Select surface(s) by using the options in the SELECT SURFACE dialog box.
- Define a local coordinate system.
- Select a coordinate plane as the mirror plane.
- Select APPLY and a eta/DYNAFORM Question dialog box appears. Select YES/NO to mirror surfaces to the ORIGINAL/CURRENT part.

### 5.2.11 SCALE SURFACE



This function allows the user to enlarge or shrink the selected surfaces by user-defined scale factors.

- Select a surface by using the options in the SELECT SURFACE dialog box.
- Define a local coordinate system.

- Enter a scale factor in the direction of U, V and W.
- After the surface is scaled, the user can select AGAIN to repeat the operation, REVERSE OPERATION to reverse the operation, or SELECT SURFACE to select a surface.

### 5.2.12 CREATE BOUNDARY LINE



This function allows the user to create a boundary line around the selected surfaces. The boundary line will be generated automatically in the current part after the surfaces are selected.

### 5.2.13 CREATE SECTION LINE



This function allows the user to create section lines on the selected surface.

- Select the surface.
- eta/DYNAFORM places edge numbers on the surface and prompts the user to enter the numbers of the section lines along edges 1 and 2.
- eta/DYNAFORM creates a set of section lines along edges 1 and 2. These lines are stored in the current part.

### 5.2.14 RESPACE U-V LINE



This function allows the user to change the density of U-V lines of the selected surface. The procedure is similar to those described in Section 5.2.13.





### 5.2.15 REVERSE NORMAL

This function allows the user to reverse the surface normal direction. The surface normal affects light source shading on some workstations. REVERSE NORMAL also controls the normal direction of the plate elements during mesh operation.

The user can select a desired surface by using the options in the SELECT SURFACE dialog box. The surface normal direction is reversed after a surface is selected.



### 5.2.16 SURFACE INTERSECT

This function allows the user to create a line at the intersection between two selected surfaces.

Select the two surfaces. An intersection line is generated automatically after the surfaces are selected. The intersect line is stored in the current part.



### 5.2.17 SURFACE SPLIT

This function is used to split a selected surface into two surfaces using a defined line/curve.

After the user selects a surface, the user can select one of the following six options to define the split line.

#### 1. TWO BOUNDARY POINTS

eta/DYNAFORM splits the selected surface once two points are picked on the boundary.

#### 2. SECTION U-V LINE

The user can select a U-V line on the selected surface using mouse cursor. The surface will split along this U-V line.

#### 3. SURFACE INTERSECTION

Once two surfaces are selected, the first surface is split at the intersection line with the

second surface.

#### 4. SPLINE CURVE

The user may pick up to 500 points on the surface to define a spline line as the splitting line.

#### 5. LINEAR SEGMENT

The user may pick up to 500 locations on the surface to define a polygon as the splitting line.

#### 6. RESPACE U-V LINES

The user is prompted to enter two values to define a new set of U-V lines.

#### 7. CURVE IN DATABASE

This function allows the user to pick an existing curve in the database to define the split surface line.

#### 8. OVERLAP SURFACE

This function allows the user to split a selected surface using an overlapping surface that is smaller than the selected surface. The first selected surface will split along the boundary of second surface.

#### 5.2.18 TRIM SURFACE



This function allows the user to trim out a portion of a selected surface. The work is completed using the TRIM SURFACE dialog box.

1. Select **SELECT SURFACE** to copy.
2. Select an existing closed curve as the trim line.
3. If no line is available in the display area, choose **EXIT**, and select **DEFINE CURVE**.

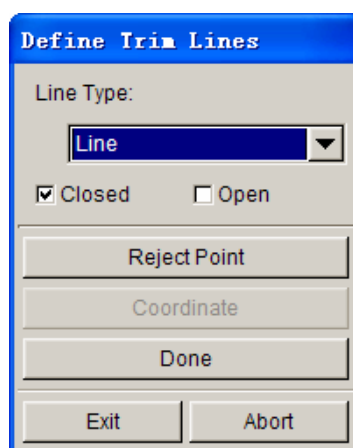


Figure 5.2.6 Define Trim Line dialog box

4. **DEFINE TRIM LINE** as illustrated in Figure 5.2.6 is displayed.
  - Select one of the curve types: line (polygon), circle, spline curve, or intersection line.
  - Select **CLOSED** or **OPEN** to close the defined line automatically or to leave it open.
  - Select points on the surface to form a line.

If the type is a **CIRCLE**, a circle will be formed and shown after two points are selected. The first point is the center, and the distance to the second point is the radius. The **COORDINATE** button then enables the user to select or define a point.

If the type is **LINE** or **SPLINE CURVE**, the user needs to pick a sequence of points and select **DONE** to complete the definition of the line.

If the type is **INTERSECTION LINE**, no point needs to be selected. The user selects another surface to define the intersection line.

During the point selection, the user can select **REJECT POINT** to deselect the last selected point.

- After the curve is defined, select **DONE**
5. After the curve is defined or selected, select **EXIT**. The eta/DYNAFORM Question dialog box, as illustrated in Figure 5.2.7 appears.

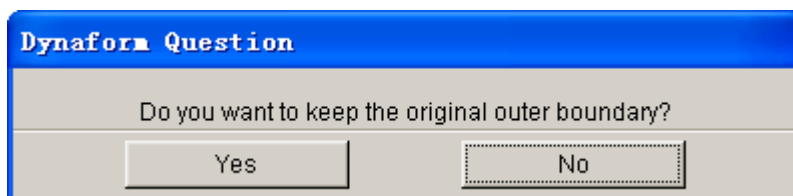


Figure 5.2.7 DYNAFORM Question dialog box

Selecting **YES** will trim out the inner boundary of the selected/defined line. Selecting **NO** will retain the inner boundary of the selected/defined line.

### 5.2.19 REMOVE HOLE

This function allows the user to remove trim holes from a selected surface. The user selects a surface by using options in the SELECT SURFACE dialog box. The holes on the surface can be removed automatically.

1. Select the **REMOVE HOLES** icon from the **PREPROCESS/SURFACE** menu.
2. Select the surface to be altered.
3. Select the type of hole; either outer boundary hole or inner hole.
4. Select a method for removing the holes, either individually or all at once.

*NOTE: For outer boundary holes, you must define two points, one on either side. eta/DYNAFORM will then fill the hole.*

### 5.2.20 SKIN SURFACE

This function allows the user to create a skin surface from a selected series of section lines. The direction of the lines must be consistent. The order of the lines must be selected from one end to the other.

### 5.2.21 UNTRIM SURFACE



This function allows user to restore the trimmed surface to the original, untrimmed surface.

### 5.2.22 DUPLICATE SURFACE CHECK



The function allows the user to check and highlight duplicate surfaces and move duplicate surfaces in a new part.

### 5.2.23 GENERATE MIDDLE SURFACE



This function enables the user to generate middle surfaces of 3D sheet-metal parts.

The following example creates a middle surface of a part with varying thickness.

1. Once this function is selected, the SELECT SURFACE dialog box appears and enables the user to select all the surfaces of the solid part. See Figure 5.2.8.

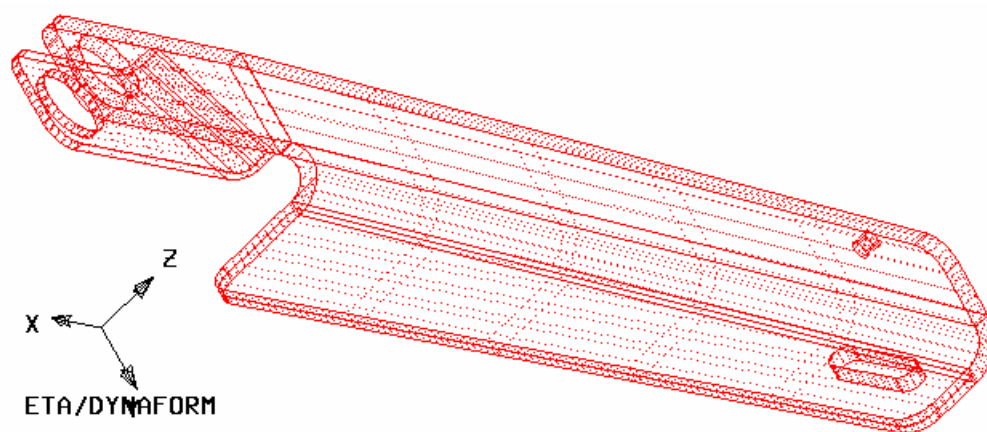


Figure 5.2.8 Select all surfaces used to generate mid surface

2. After selecting the entire surfaces, the program will generate the mid surface automatically as Figure 5.2.9 shown. At the same time a new part named MIDSRF is created and all the created

mid surfaces are placed in this part.

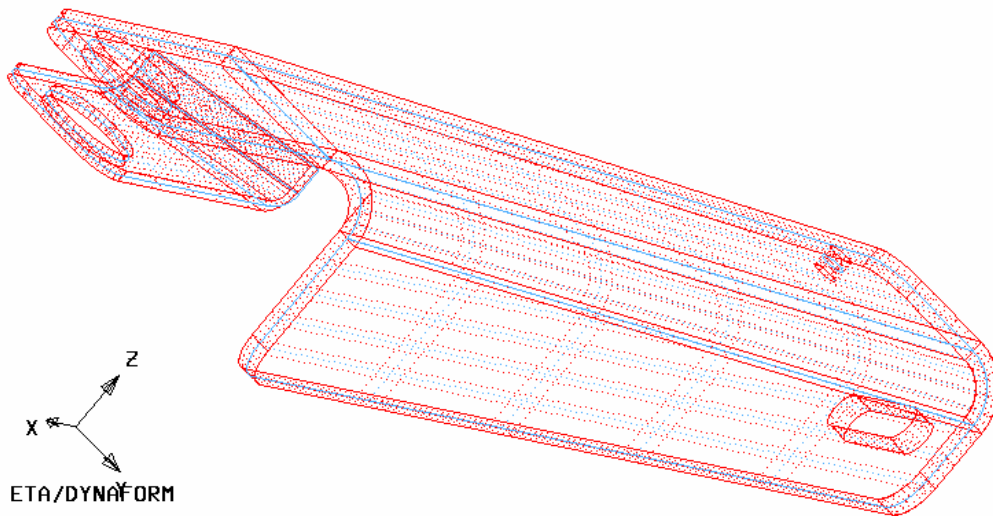


Figure 5.2.9 Mid Surfaces

3. After the mid surfaces are created, the SELECT OPTION dialog box as illustrated in Figure 5.2.10 shown will pop-up.

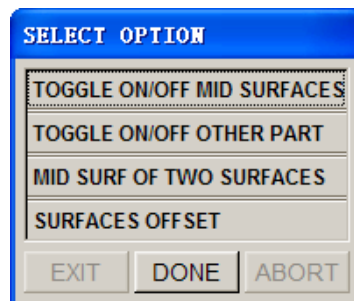


Figure 5.2.10 Select Options dialog box

### **TOGGLE ON/OFF MID SURFACES**

This function enables user to toggle on/off the MIDSURF part. Figure 5.2.11 shows the mid surface only.

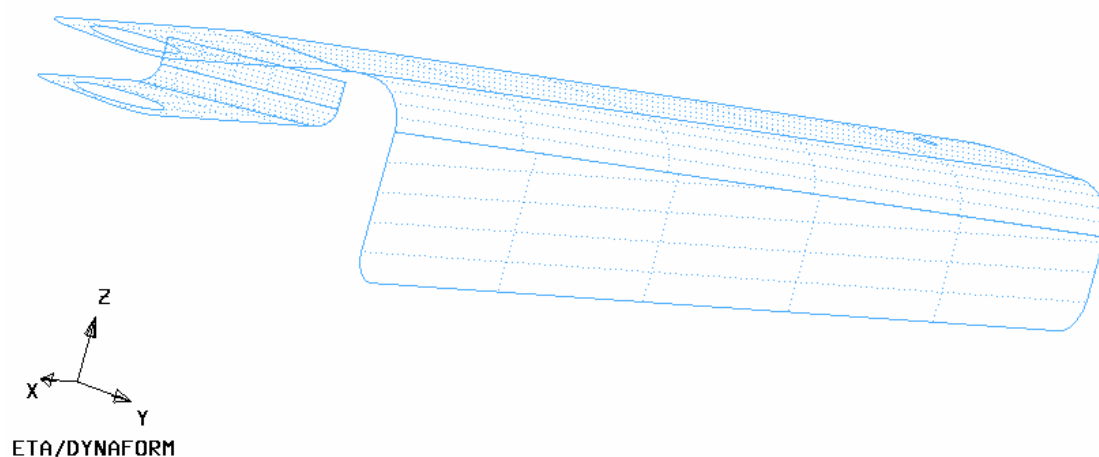


Figure 5.2.11 Middle surfaces display only

**TOGGLE ON/OFF OTHER PART**

This function enables user to view only the MIDSURF part by hiding all other parts.

**MID SURF OF TWO SURFACES**

This function enables user to create the mid surface by manually selecting a pair of surfaces. For some complicated models, it is very difficult to get the satisfactory result. Sometimes the program may miss to create the mid surfaces due to poor surface quality. The user can use this function to create the missed mid surface manually.

After the function is selected, the Surface Select dialog box is displayed. User needs to select a pair of surface and then clicks OK button. The mid surface of selected surfaces will be created. Please refer to Figure 5.2.12.

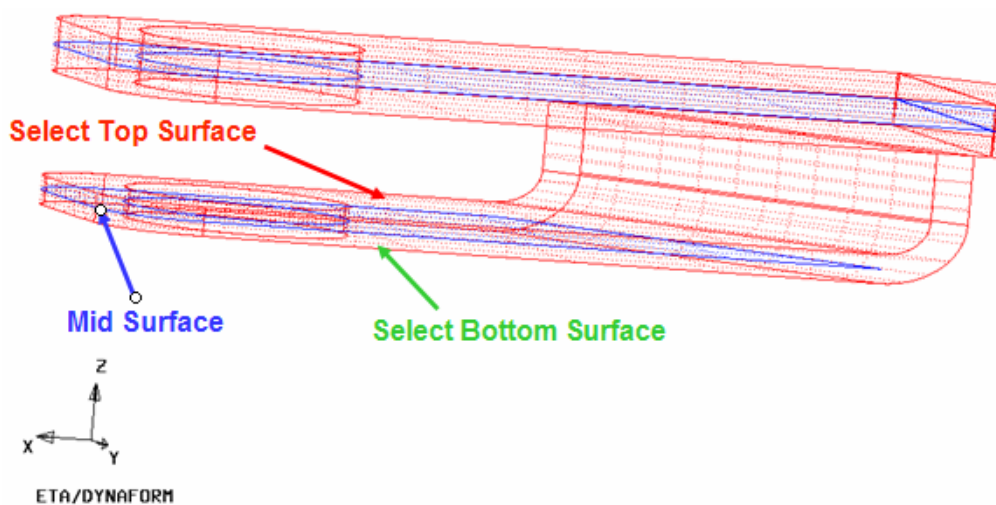


Figure 5.2.12 mid surface of two surfaces

**SURFACE OFFSET**

This function enables user to create the mid surface through offsetting one surface layer, either top or bottom layer. Sometimes some surfaces of the models are lost after imported to eta/DYNAFORM database, hence the program can't automatically create the mid surface. The user can use this function to offset one surface layer to create the mid surface.

After the function is selected, the Surface Select dialog box is displayed. The user can select the surface needed to be offset. The program will then display the SELECT DIRECTION dialog box as illustrated in Figure 5.2.13. The user can click the REVERSE option to reverse the offset direction or click the ACCEPT to accept the existing offset direction.

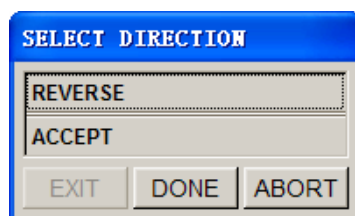


Figure 5.2.13 Select Offset direction dialog box

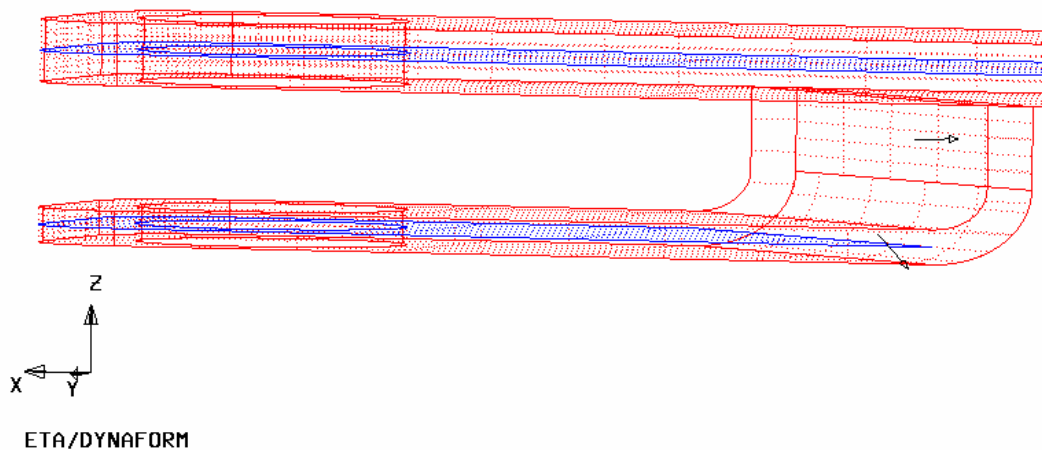


Figure 5.2.14 Surface Offset Direction

After selecting the offset direction, the new offset surface will be created. Please see the Figure 5.2.15.



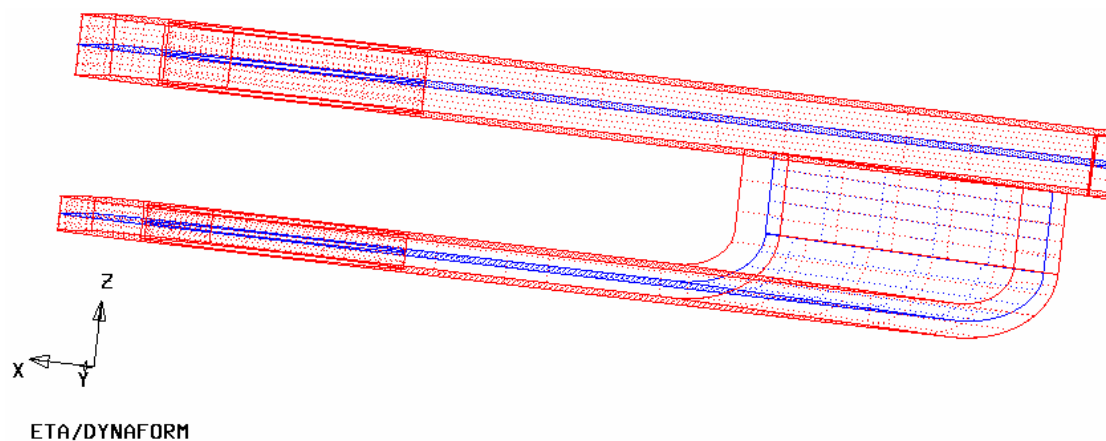


Figure 5.2.15 Create mid surface by offset

5.2.24 GROUP SURFACE



This function is used to group the top and bottom surface layers of sheet metal parts. The GROUP SURFACE operation is similar to the GENERATE MIDDLE SURFACE operation. The following steps are used to group top and bottom surface layers.

1. Once the function is selected, the SELECT SURFACE dialog box appears to prompt the user to select all the surfaces that used to be grouped.
2. After selecting entire surfaces, the program will group the top surface automatically. If some top surfaces are not selected, the user can click the ADD TOP SURFACES option to add top surfaces. Refer to the Figure 5.2.16.

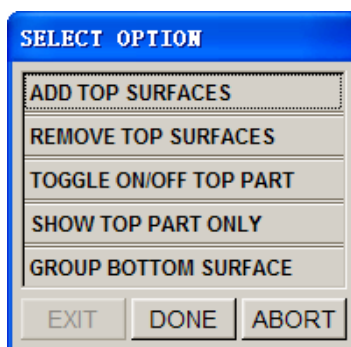


Figure 5.2.16 Group Options dialog box

3. User can click GROUP TOP SURFACE to group all top surfaces as illustrated in Figure 5.2.17.

The grouped surfaces are highlighted, while the SELECT OPTION dialog box as illustrated in Figure 5.2.17, is displayed.

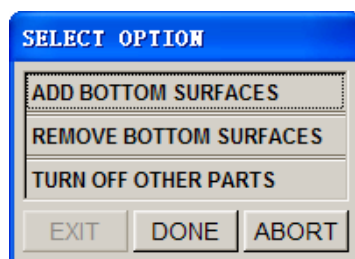


Figure5.2.17 Select Option dialog box

4. The bottom surfaces are highlighted. If the result is not satisfactory, the user can select ADD BOTTOM SURFACES or REMOVE BOTTOM SURFACES option to adjust the result.
5. Click on DONE button to group the bottom surfaces of the part (Figure 5.2.18). At this point, Figure 5.2.15 is displayed.

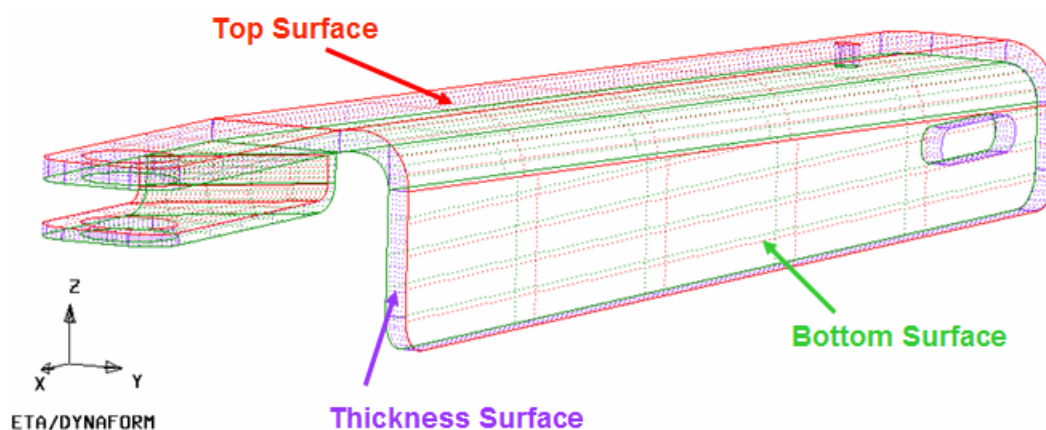


Figure 5.2.18 Group Bottom Surfaces

The thickness surfaces are put in the part called THKSRF1, the top surfaces are put in the part called TOPSRF1, and the bottom surfaces are put in the part called BOTSRF1. The number "1" will be incremented by 1 for each subsequent grouping operation.

### 5.3 ELEMENT (CTRL+E)

The ELEMENT menu contains the functions to generate and modify elements. The functions are shown in Figure 5.3.1.



Figure 5.3.1 Element Options Menu

- **LABEL ELEMENT ON**  
Toggles ON/OFF to show/hide the element labels.
- **SHRINK ELEMENT ON**  
Reduces the size of elements by 20%.

A detailed description of the other functions in the ELEMENT menu is given in the following sections.

### 5.3.1 2 LINE MESH



The created elements will be stored in the current part.

- Select two lines in the submenu window. Refer to Section 5.2.1 for available options.
- Enter the integers in N1, N2, N3, and N4 fields in the displayed dialog box.

Integer N1 is the number of elements created along line 1. Integer 2 is the number of elements created along the side connecting line 2 at the starting end of line 1. Integers N3 and N4 are the number of elements created along line 2 and the side between the two selected lines. These four integers must meet the following criteria:

$$N1 < 2N3, N3 < 2N1, N2 < 2N4, \text{ and } N4 < 2N2$$

If N3 and N4 are omitted, N1 is the number of elements along the selected lines, and N2 will be the number of elements between the two lines.

- Select a button in the eta/DYNAFORM Question dialog box as illustrated in Figure 5.3.2.

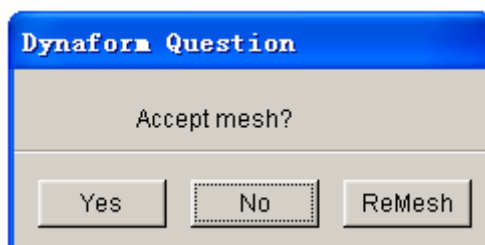


Figure 5.3.2 DYNAFORM Question dialog box

Select YES to accept the mesh, and the program will prompt for more lines for another 2L mesh.

Select NO to reject the mesh, and the program will prompt for more lines for another 2L mesh.

Select REMESH to reject the mesh, and the program will prompt for new integers, N1, N2, N3, and N4.

### 5.3.2 3 LINE MESH



This function allows the user to generate elements in an area defined by three lines. After all of the lines are selected, the submenu SELECT OPTION dialog box, as illustrated in Figure 5.3.3 is displayed.

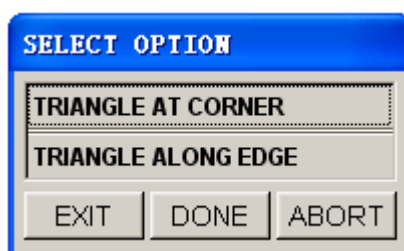


Figure 5.3.3 Select Option dialog box

- **TRIANGLE AT CORNER**  
The user needs to input two numbers, N1 and N2, in the displayed dialog box. The third number will be the N1 element along each line of line 1 and line 2. N2 elements will be along line 3. The triangular elements will be generated at the vertex of line 1 and line 2.
- **TRIANGLE ALONG EDGE**  
The user needs to input only one number, N, in the displayed dialog box. There is N elements along each of the three lines. All triangular elements will be along the third selected line.

### 5.3.3 4 LINE MESH



The operation in this function is similar to 2 LINE MESH.

### 5.3.4 SURFACE MESH



This function allows the user to create auto-mesh on the selected surface(s). The dialog box is illustrated in Figure 5.3.4.

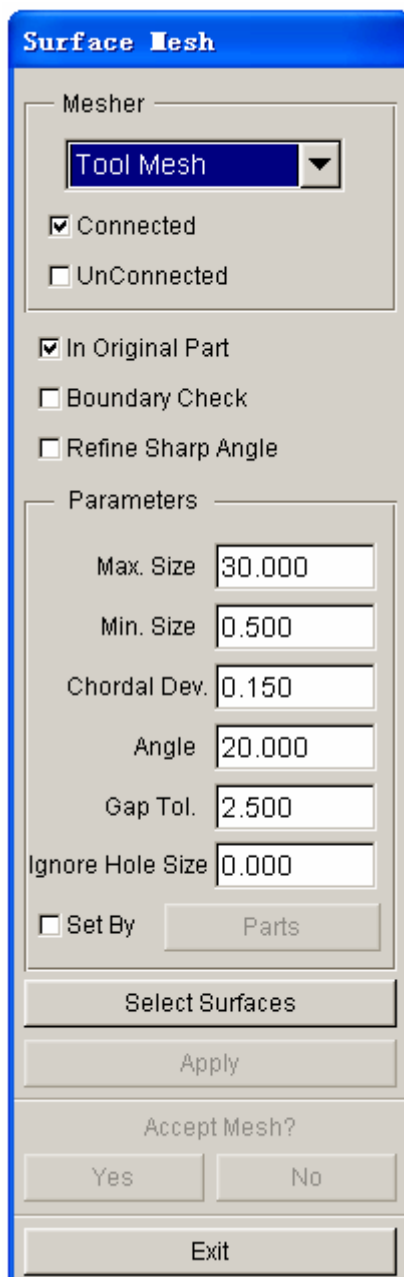


Figure 5.3.4 Surface Mesh dialog box

- Select the SELECT SURFACES button. Refer to Section 4.4.3, PART/ADD SURFACE, for more information.
- Select a type of tool: CONNECTED or DISCONNECTED MESH, PART MESHER, or TRIANGLE MESHER.

CONNECTED TOOL MESH will connect the adjacent surfaces if the gaps are within the surface gap tolerance (GAP TOL).

DISCONNECTED TOOL MESH will leave gaps between elements that are located on adjacent surfaces.

PART MESHER is the new topology mesher.

TRIANGLE MESHER will only generate triangle elements.

- Toggle on/off **IN ORIGINAL PART**. This option includes the created elements in original part or in current part.
- Toggle on/off **BOUNDARY CHECK**. This option checks and displays the model boundary after building the topological structures.
- Toggle on/off **REFINE SHARP ANGLE**. This option checks and repairs triangular element with very sharp angle (less than 5 degrees) due to bad quality of the CAD surface. The meshing speed will be reduced by 40 to 50% when this option is selected. Therefore, it should be used only when the sharp angled elements are encountered in the mesh.
- Enter the numbers of **MAXIMUM SIZE**, **MINIMUM SIZE**, **ANGLE** and **CHORDAL DEVIATION** in the data field.

#### **MAXIMUM SIZE**

Controls the maximum element size.

#### **MINIMUM SIZE**

Controls the minimum element size.

#### **ANGLE**

Controls the inclination of the adjacent elements.

#### **CHORDAL**

Controls the number of elements at the radius.

#### **IGNORE HOLE SIZE**

Fills interior holes of the surface with mesh if the hole size is less than the input criteria.

- Select **APPLY** to mesh the surface

If TOOL MESH is selected:

eta/DYNAFORM will create mesh suitable for punch or die definition on the selected surfaces. If the number of selected surfaces is equal to or greater than 2, the BOUNDARY CHECK is activated. The function first builds topological structures and then displays the STATUS WINDOW. See Figure 5.3.5. A message will be printed in the message prompt window.

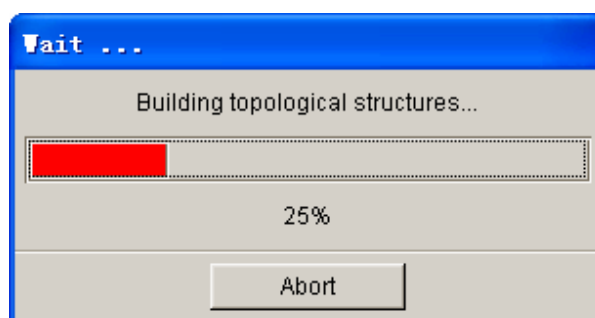


Figure 5.3.5 Building Topological Structures

- After building the topological structures, the eta/DYNAFORM Question dialog box, as illustrated in Figure 5.3.6 is displayed. Select YES to accept the boundary and continue to the next step. Select NO to display the BOUNDARY CHECK dialog box as illustrated in Figure 5.3.7.

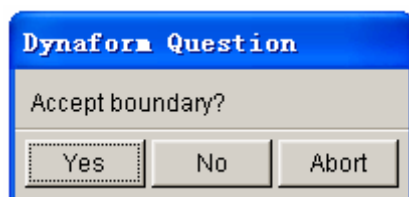


Figure 5.3.6 DYNAFORM Question

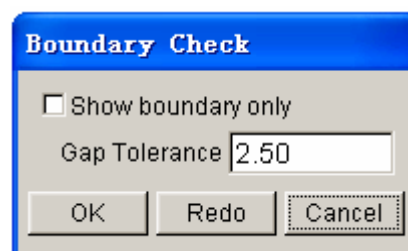


Figure 5.3.7 Boundary Check

Select/deselect SHOW BOUNDARY ONLY to display surfaces with model boundaries. Select OK to accept the gap tolerance. Select REDO to enter a new gap tolerance to rebuild the topological structures.

- After the user selects or accepts the gap tolerance, eta/DYNAFORM begins to mesh the surface boundaries. The status windows for meshing surface boundary and mesh generation are illustrated in Figure 5.3.8 and Figure 5.3.9, respectively.



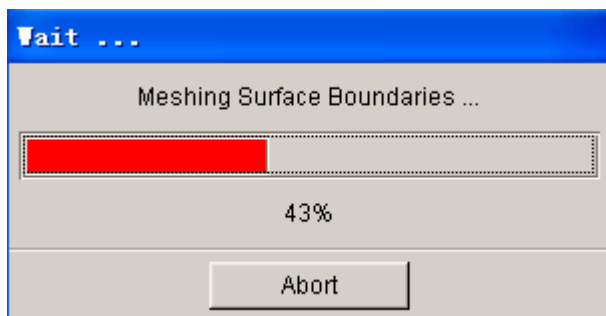


Figure 5.3.8 Meshing Surface Boundaries

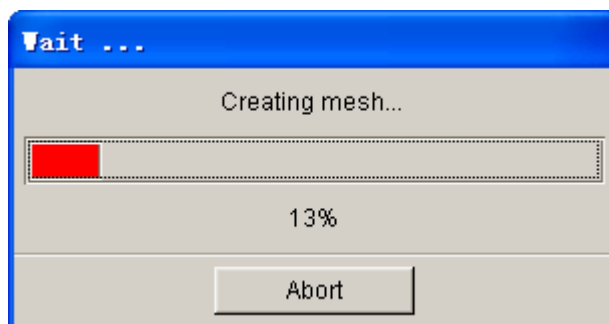


Figure 5.3.9 Creating Mesh

- The ACCEPT MESH CONFIRMATION (YES and NO) field is enabled in the SURFACE MESH dialog box. Select YES or NO to accept or reject the created mesh.
- Select OK to exit.

**NOTE:** When any status window appears, the user can abort the current operation by selecting **ABORT**.

If PART MESHER is selected:

PART MESHER is the new topology mesher used to generate part mesh for MSTEP analysis. Refer to Figure 5.3.10 for the list of control parameters.

- CHECK SURFACE  
Checks for duplicate surfaces prior to mesh operation.
- MESH BY PART  
Mesh parts one at a time. The mesh will not be connected between parts.

- **AUTO REPAIR**  
Automatically merges gaps between elements and removes free nodes from database.
- **IGNORE HOLE SIZE**  
Fills an interior hole in the surface with mesh if the hole size is less than the input criteria.
- **MESH QUALITY**  
Refer to Figure 5.3.11 for mesh quality control parameters.

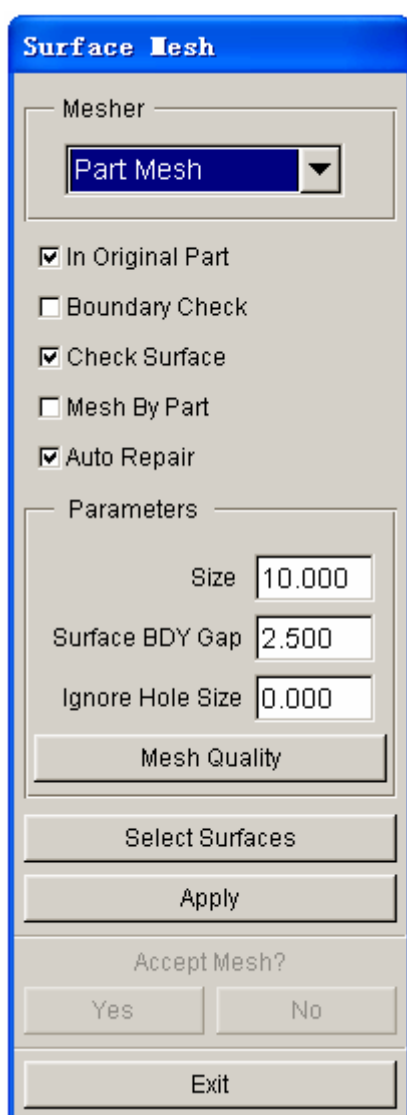


Figure 5.3.10 Part Mesher dialog box

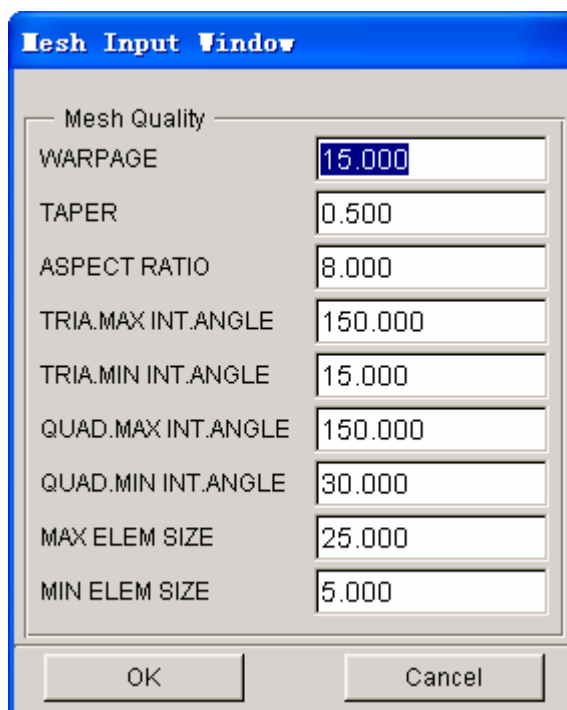


Figure 5.3.11 Mesh Quality dialog box

If TRIANGLE MESHER is selected:

The user can enter only the element size and the gap tolerance as shown in Figure 5.3.12.

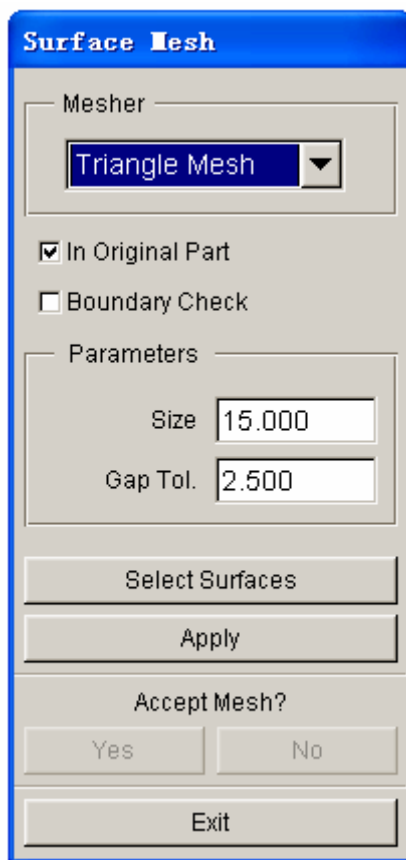


Figure 5.3.12 Triangle Mesher dialog box

### 5.3.5 LINE MESH



This function allows the user to generate one-dimensional elements (beam/rod or plotel) automatically along a selected line.

- Enter the size or number of elements to be created in the Element Control dialog box as illustrated in Figure 5.3.13. Then, click on OK button.

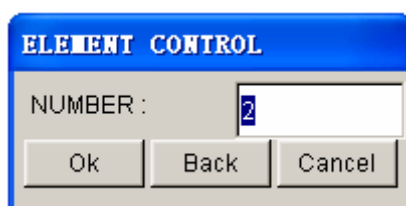


Figure 5.3.13 Element size

**NOTE:** Mesh pattern can be controlled by size or number. This option is defined in Option/Mesh Control/Line Mesh Method.

- Select a line.
- Choose either NODE/POINT or ORIENTATION VECTOR from the dialog box as illustrated in Figure 5.3.14.



Figure 5.3.14 Control Keys

- If NODE/POINT is selected, the dialog box as illustrated in Figure 5.3.15 is displayed. This window prompts the user to select the appropriate NODES/POINTS and also provides an option to define the orientation vector.
- If the user selects ORIENTATION VECTOR, the dialog box as illustrated in Figure 5.3.16 is displayed. This window prompts the user to define the orientation vector.
- Select YES/NO/REMESH from the mesh confirmation window to accept, reject, or remesh the result.

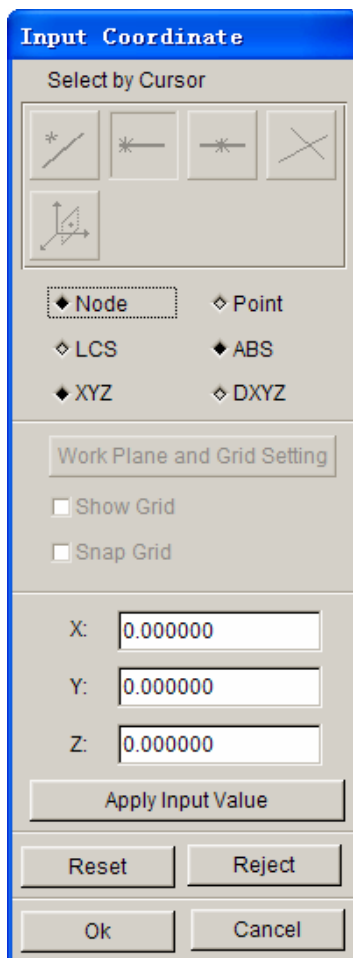


Figure 5.3.15 Input Coordinate dialog box

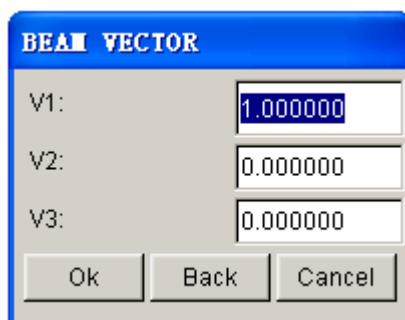


Figure 5.3.16 Beam Vector dialog box

**5.3.6 2 LINE POINT MESH**



This function allows the user to generate a mesh between two lines based on the points of the selected lines.

- Select two lines.
- Enter the number of layers between the two lines.
- Confirm acceptance of the generated mesh.



### 5.3.7 DRAG MESH

This function generates elements based on the selected element pattern. One-dimensional elements will be dragged to form plate elements, and two-dimensional elements (plate) will be dragged to form solid elements (prism).

**NOTE:** *In eta/DYNAFORM, only the same types of elements can be included in a part. For example, if the current part contains plate (shell) elements, the user cannot create any solid elements in this part. eta/DYNAFORM will display a warning message in the prompt window to alert the user.*

#### 1. ONE LINE DRAG

- Select the elements for the drag pattern by using the options in the SELECT ELEMENT dialog box. Refer to Figure 4.4.2 of Section 4.3.2, PART/ADD ELEMENT.
- Select a line. If the number of points is  $n$ ,  $n-1$  elements will be generated for each pattern element.
- The user can accept, reject, or remesh.

#### 2. NORMAL DRAG

- Select the plate element pattern by using options in the SELECT ELEMENT dialog box. Enter the thickness of normal drag and the number of layers in the POP-UP dialog box.

- The user can accept, reject, or remesh.

**NOTE:** *Plate element normals should be consistent in the drag pattern prior to the execution of NORMAL DRAG.*

### 3. 3 OR 4 LINE DRAG

- Select the plate element pattern by using the options in the SELECT ELEMENT dialog box.
- Select three or four control lines to form a volume with the drag pattern. The user must select the control lines in clockwise or counterclockwise order.
- Enter the number of layers.
- The user can accept, reject, or re-mesh.

### 5.3.8 CREATE ELEMENT



This menu allows the user to create elements by connecting selected nodes or points. The supported element types are compatible with LS-DYNA. The element types are shown in Figure 5.3.17.

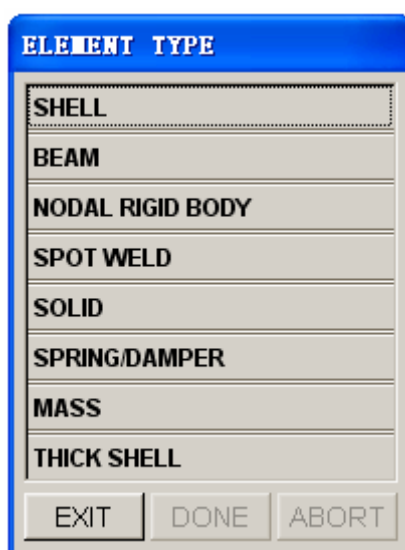


Figure 5.3.17 Element Type dialog box

## 1. BEAM

- Select two points or nodes by cursor in the screen.
- Define the material axis by selecting a third node/point or by defining the orientation vector.

BY NODE

BY POINT

BY VECTOR

Up to three values may be entered in the POP-UP dialog box to define a vector in the beam local XY plane. The local Y-axis of the beam will be plotted at the first node of the bar element as it is generated.

## 2. NODAL RIGID BODY

The user selects nodes or points in the INPUT COORDINATE dialog box. The NODAL RIGID BODY is created automatically in the current part once DONE is selected.

## 3. SPOT WELD

A SPOT WELD is created automatically in the current part after two nodes or points are selected in the INPUT COORDINATE dialog box.

## 4. SHELL

This function allows the user to create quadrilateral and triangular elements in the current part. The INPUT COORDINATE dialog box will appear to enable the user to select a node/point to define the element.

- **CREATE QUADRILATERAL ELEMENT**

The element is created automatically once four nodes or points are selected.

- **CREATE TRIANGULAR ELEMENT**

The user must click OK or the middle mouse button to complete the element after three nodes or points are selected.

## 5. SOLID

This function allows the user to create cube, wedge, and tetrahedron elements by using options in the INPUT COORDINATE dialog box.



- **TETRAHEDRON**

Select four nodes/points, and select OK, or click the middle mouse button. The first three nodes define the base of the tetrahedron. They should follow the right hand rule to point to the fourth node.

- **PENTAHEDRON**

Select six nodes/points (pick the second set of three nodes/points in the same order as the first set), and select OK, or click the middle mouse button. The first three nodes define the bottom face of the pentahedron. They should follow the right hand rule to point to the top face.

- **HEXAHEDRON**

Select eight nodes/points (pick the second set of four nodes/points in the same order as the first set) and the element will be created automatically. The first four nodes define the bottom face of the hexahedron. They should follow the right hand rule to point to the top face.

6. **SPRING/DAMPER (2 NODE ELEMENTS)**

The element is created after two nodes/points are selected.

7. **MASS (1 NODE ELEMENT)**

- Enter the value of the mass in the field of the displayed dialog box.
- Select nodes/points as the mass element by using options in the INPUT COORDINATE dialog box.

8. **THICK SHELL**

The element is created after eight nodes/points are selected (the second set of four nodes/points is in the same order of the first set). The first four nodes define the bottom face of the thick shell. They should follow the right hand rule to point to the top face.

### 5.3.9 COARSE ELEMENT



This function allows the user to coarsen two or four QUAD elements into one QUAD element, two

TRIA elements into one QUAD element, or a QUAD/TRIA combination into one QUAD element.

The user selects an element by using the options in the SELECT ELEMENT dialog box. Once the element is selected, a larger element will be created.

- Select two adjacent plate elements, and select OK in the SELECT ELEMENT dialog box, or click the middle mouse button. If 4 adjoining QUAD elements are selected, the program will combine them into one quad element.
- Repeat the previous step in order to coarsen more elements.
- Select DONE in the SELECT ELEMENT dialog box. Each of the selected pairs above will be combined into one element.

### 5.3.10 SPLIT ELEMENT



This function divides any SHELL elements into multiple elements based on the options described below. The dialog box is illustrated in Figure 5.3.18.

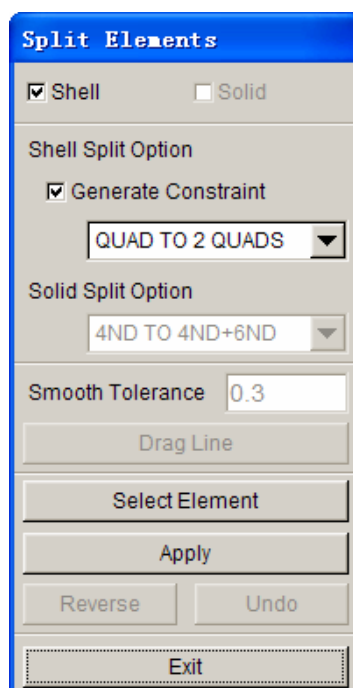


Figure 5.3.18 Split Element dialog box

### 1. SHELL SPLIT OPTIONS

- Generate Constraint enables the user to create the constraint on the new middle node after split the element.
- Click the arrow next to SHELL SPLIT OPTION to access the drop-down menu of options as illustrated in Figure 5.3.19.

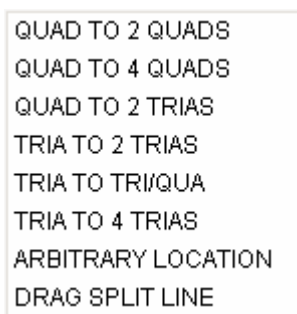


Figure 5.3.19 Shell Split Options

#### **QUAD TO 2 QUADS**

Split a quadrilateral element into two quadrilateral elements.

#### **QUAD TO 4 QUADS**

Split a quadrilateral element into four quadrilateral elements.

#### **QUAD TO 2 TRIAS**

Split a quadrilateral element into two triangular elements.

#### **TRIA TO 2 TRIAS**

Split a triangular element into two triangular elements.

#### **TRIA TO TRI/QUA**

Split a triangular element into a triangular and a quadrilateral element.

#### **TRIA TO 4 TRIAS**

Split a triangular element into four triangular elements.

### ARBITRARY LOCATION

First select an element, and then select two nodes/points arbitrarily on the boundary of the element.

### DRAG SPLIT LINE

The user can select DRAGLINE to split several elements and enter a number into the field to change the smooth tolerance.

- Select SELECT SPLIT OPTION for splitting.
- Click SELECT ELEMENTS to open the SELECT ELEMENT dialog box.
- Select the elements, and then select DONE to go back to the SPLIT ELEMENT dialog box.
- Select APPLY to split the selected elements. After the elements are split, the user can select UNDO to cancel the operation or REVERSE to change the split direction.

### 5.3.11 PROJECT ELEMENT



This function is used to project all elements in a part onto an existing mesh of another part. The project direction is the W-direction of a defined LCS.

- Select the target part by using options in the SELECT PART dialog box.
- Define a coordinate system by using options in the LCS dialog box.
- Select a source part in order to project the element on the source part onto the target part.

**NOTE:** *eta/DYNAFORM will automatically split the source mesh at the elements that have no projection point.*

### 5.3.12 REVERSE ELEMENT



This function allows the user to reverse the element normal of selected elements.

The user selects elements by using the options in the SELECT ELEMENT dialog box. The orientations of the selected elements are reversed when the selection is complete.

### 5.3.13 MIRROR ELEMENT



This function allows the user to mirror elements based on a defined LCS and mirror plane. The Mirror Elements dialog box is illustrated in Figure 5.3.20

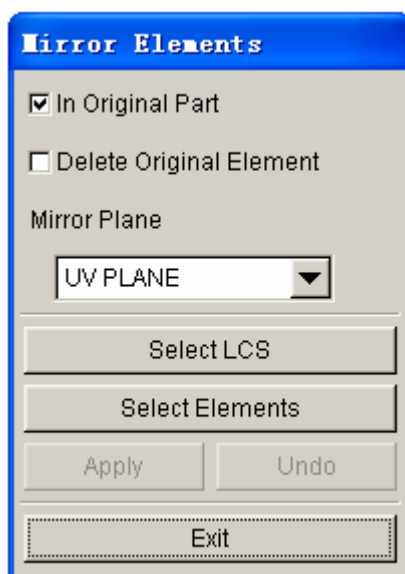


Figure 5.3.20 Mirror Element dialog box

For more information on MIRROR ELEMENTS, refer to Section 5.1.11, MIRROR LINES.

### 5.3.14 COPY ELEMENT



This function allows the user to generate a set of duplicate elements. The user may simultaneously

translate, rotate, offset (in the normal direction), or map the copied elements. The dialog box is illustrated in Figure 5.3.21. Refer Section 5.1.5 for COPY and TRANSFORM LINE and Section 10.2.7 for MATING TOOLS FOR OFFSETTING.

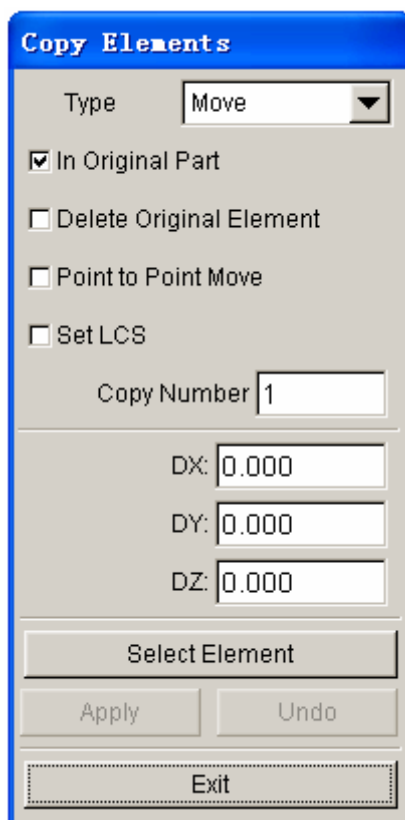


Figure 5.3.21 Copy Element dialog box

For the MAPPING option, the user defines only two coordinate systems. The selected elements are copied from the place in the first coordinate system to the same place in the second coordinate system.

### 5.3.15 MODIFY ELEMENT



This function allows the user to edit a selected element. The modified element will have the same element ID as the original element.

### 5.3.16 CHANGE ELEMENT NUMBER



This function is used to change the ID number of the selected element.

### 5.3.17 RENUMBER ELEMENT



This function allows the user to renumber all the elements in the database. There are two options available: SEQUENCE or BY PART.

#### SEQUENCE

The user inputs the starting number in the POP-UP dialog box (default is 1). The numbers of all the elements are renumbered sequentially from the starting number.

#### BY PART

eta/DYNAFORM Question dialog box as illustrated in Figure 5.3.22 appears.

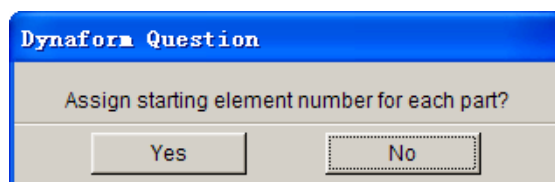


Figure 5.3.22 Starting Element Number Question

#### 1. YES

eta/DYNAFORM prompts:

***ENTER STARTING ELEMENT NUMBER FOR PART: xxx***

and displays a dialog box for the user to enter the starting element number of this part.

Once a number has been entered, the program will confirm with the following prompt:

***ELEM x TO xx ASSIGNED, NEXT ELEM NO: yyyy***

This procedure will be repeated for every part in the database.

#### 2. NO

Two INPUT dialog box will appear. The first is for entering the starting element number, and the second is for entering the element increment between parts (default is 1000).

### 5.3.18 DELETE ELEMENT



This function allows the user to remove selected elements from the model.

The SELECT ELEMENT dialog box appears and the selected elements are deleted after the user selects OK.

### 5.3.19 IDENTIFY ELEMENT



This function allows the user to identify an element and its nodes by cursor selection. The number of selected elements and its nodes will be highlighted. A detailed message in the message prompt window will display the element number, part name, the number to which it is assigned, and the node numbers that it contains.

### 5.3.20 FIND ELEMENT



This function allows the user to find and identify an element by entering its element number.

- Once the element number is entered, the program will highlight the element on the screen and label its node numbers. The node numbers will also be displayed in the PROMPT window.
- If the element is not displayed on the screen, the prompt will read:

***ELEMENT xxx IN TURNED OFF PART: (the part name)***

- If the element does not exist in the database, the prompt will read:

***ELEMENT xxxx NOT FOUND IN DATABASE***



### 5.3.21 MESH REPAIR



This function automatically repairs gaps and improves element quality in the mesh by merging nearby nodes and modifying elements. Caution should be taken when this function selected as it can change the geometry of the tooling. Several user-defined controls are given for identifying and repairing the inferior mesh. After selecting elements, the REPAIR PARAMETERS dialog box as illustrated in Figure 5.3.23 is displayed.

**Input Parameters**

Priority

4 Tria     2 Tria

Feature L     Warpage

Repair:

Short Edge

Sharp Angle

AND     OR

DTOL: 0.2

ATOL: 10.0

CA\_TOL: 150.0

W\_TOL: 15.0

F\_TOL: 30.0

GAP\_TOL: 0.1

OK    Back    Cancel

Figure 5.3.23 Mesh Repair dialog box

#### **PRIORITY**

The user can set the priority to split QUADs into 2 or 4 Triangles, and take FEATURE LINE or WARPAGE into account when doing the repair.

#### **REPAIR**

The user can select SHORT EDGE and/or SHAPE EDGE or neither.

### PARAMETERS FOR CHECKING BAD ELEMENTS

These parameters are the criteria for repairing bad elements: MINIMUM ELEMENT SIZE (DTOL), SHARP ANGLE TOLERANCE (ATOL), LARGE ANGLE TOLERANCE (CA\_TOL), WARPAGE TOLERANCE (W\_TOL), FEATURE LINE ANGLE TOLERANCE (F\_TOL), and GAP TOLERANCE (GAP\_TOL).

After the necessary parameters are defined, click OK to perform mesh repair. The selected mesh will be checked and repaired according to the defined criteria. The program will display the repaired mesh and prompt for acceptance.

## 5.4 NODE (CTRL+N)

In eta/DYNAFORM nodes and points as "points" in a three dimensional space are different entities. Points are used to form lines and surfaces while nodes are used to form elements. There are two types of nodes, referenced nodes and un-referenced nodes (free nodes). A referenced node is a node that is used by elements and is represented by a dot. An un-referenced (free) node is a node that is not used by any element and is represented by an asterisk. The functions in this menu are used to create nodes, change the position of the nodes, and delete free nodes. The Node dialog box is illustrated in Figure 5.4.1.



Figure 5.4.1 Node dialog box

- **LABEL NODES ON**

Toggles the node labels ON/OFF.

A detailed description of each function is given in the following sections.

#### 5.4.1 CREATE NODE



This function allows the user to create un-referenced nodes at selected points or by entering coordinates. Refer to Section 5.1.1 for details on how to select points or enter coordinates. The user can create nodes by using options in the INPUT COORDINATE dialog box.

#### 5.4.2 ADD NODES BETWEEN TWO NODES/POINTS



This function is used to generate nodes that are spaced equally between two existing nodes or points. The new nodes are displayed as un-referenced (free) nodes.

The user selects two nodes/points by using options in the INPUT COORDINATE dialog box. Then, key in the number of nodes in the input field of INPUT QUANTITY dialog box and click on the OK button to complete the operation.

#### 5.4.3 COPY NODE



This function allows the user to generate a new set of un-referenced nodes from user-specified nodal locations.

- Select nodes by using options in the SELECT NODE dialog box as illustrated in Figure 5.4.2.

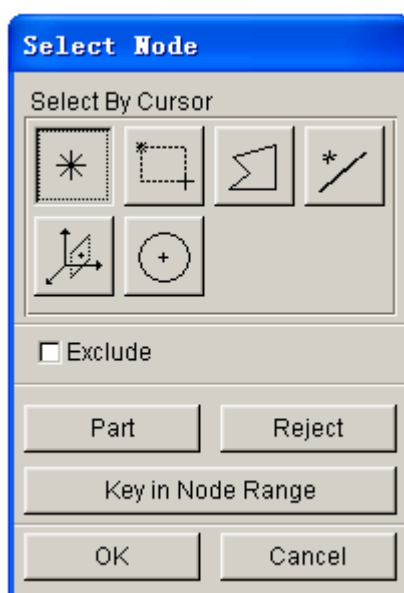








Figure 5.4.2 Select Node dialog box

Selection of nodes can be done using the following approaches:

-  By cursor.
-  By drag window. Select two diagonal points in the screen to define a window.
-  By multi-point region. Select several points on the screen to define a multi-point region.
-  Near a line. Select a line and enter the tolerance.
-  Near a plane. Select a plane and enter the location and tolerance.
-  All the nodes inside the circle will be selected.

#### **BY PART NAME**

Select the PART option and select the part that contains the nodes.

#### **BY NODE RANGE**

Select KEY IN NODE RANGE. A dialog box is displayed for input of the starting nodal number, ending nodal number, and increment.

- Select OK after node selection.

- Define a coordinate system by using options in the LCS dialog box.
- Enter the number of copies in the following displayed dialog box.

There are two ways to copy nodes:

1. MOVE  
Enter displacements in the following displayed dialog box.
2. ROTATE  
Enter the angle increment in the following displayed dialog box. The nodes will be rotated along the W-axis of the defined LCS.

#### 5.4.4 DELETE UNREFERENCED NODES



This function allows the user to remove either all or selected un-referenced nodes in the database. The un-referenced nodes are labeled with asterisks (\*).

##### **ALL FREE NODES**

All un-referenced nodes will be removed from the database.

##### **SELECTED FREE NODES**

Select free nodes by using options in the SELECT NODE dialog box.

#### 5.4.5 TRANSFORM NODE



This function allows the user to translate or rotate the selected nodes to a new location.

- Define a new coordinate system by using options in the LCS dialog box.
- Select the MOVE or ROTATE option and input the value of displacement or angle increment.
- Select nodes by using the options in the SELECT NODE dialog box.

- After the selected nodes are transformed, the user can select the following choices:
  1. **AGAIN**  
Repeats the translation or rotation.
  2. **REVERSE OPERATION**  
Performs the translation in the opposite direction.
  3. **RESELECT NODE**  
Selects a new set of nodes for another copy operation.

#### 5.4.6 MOVE NODE



This function allows the user to move nodes to a new location in the display area.

- The user selects a node by cursor.  
If a coincident node is selected, a message will be printed in the message prompt window:

***MULTIPLE NODES FOUND  
SELECT ELEMENT FOR NODE***

If the selected element is not connected with the node, eta/DYNAFORM will print the following warning message in the message prompt window:

***SELECTED ELEM IS NOT CONNECTED TO DUPL NODE  
SELECT ELEMENT FOR NODE***

- After the desired node is selected, the user selects a new location for the node by using options provided in the INPUT COORDINATE dialog box. The node is then moved to the selected location.
- **UNDO LAST**  
This is a valid selection only if a node has been moved.

#### 5.4.7 ALIGN



This function allows the user to align any selected nodes on a selected line.

- Select nodes to be aligned.

- Select a line to align the selected nodes.
- Nodes are positioned on the selected line according to the shortest distance to the line.

#### 5.4.8 SCALE NODE



This function allows the user to move nodes by scaling their coordinates.

- Select a set of nodes using options in the SELECT NODE dialog box.
- Define a coordinate system using options in the LCS dialog box.
- Enter the X, Y, Z scale factor in the next dialog window. The coordinates of the selected nodes will be scaled according to the defined scale factor.

#### 5.4.9 PROJECT NODE



This function allows the user to project a node or group of nodes onto a plane, surface(s) or a set of elements using the options described below.

##### 1. ON F. E. MESH

- Select a set of elements by using options provided in the SELECT ELEMENT dialog box.
- Create a new local coordinate system to define the projection vector. The nodes are projected along the W-axis of the defined LCS.
- Select nodes.
- Use UNDO LAST to reject the last projection.

##### 2. ON LOCAL UV-PLANE

- Create a new coordinate system to determine the UV-plane using the options provided in the LCS dialog box.

- A eta/DYNAFORM Question dialog box as illustrated in Figure 5.4.3 is displayed.

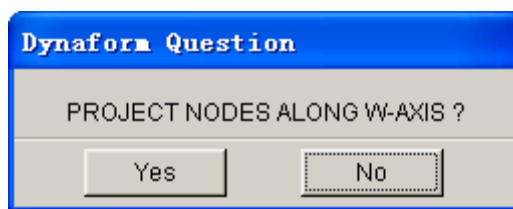


Figure 5.4.3 Project Node Question dialog box

Selecting YES enables the user to select nodes that will be projected on the local U-V plane using options in the SELECT NODE dialog box.

If NO is selected, the user has to create a new coordinate system.

### 3. ON SURFACE

The user selects a set of surfaces by using the options in the SELECT SURFACE dialog box. The selected nodes will be projected onto the selected surfaces along the projection direction.

### 4. ON SHELL ELEMENT (NORMAL)

The user selects a set of shell elements by using the options in the SELECT ELEMENTS dialog box. The selected nodes will be projected on the selected shell elements along the shell element normal.

## 5.4.10 CHECK DUPLICATE NODES



This function allows the user to renumber any duplicated node ID numbers in the database. The user cannot create duplicated node numbers in eta/DYNAFORM. However, the imported model files from other programs may contain duplicated node ID numbers. eta/DYNAFORM will renumber the duplicated node ID from the existing maximum node ID number.

If any duplicated nodes are found, the following messages will be printed in the message prompt window and in the DYNAFORM.msg message file located in the work directory:

***CHECKING nnnn NODES FOR DUPLICATE NODES***

***DUPLICATE NODE xxxx CHANGED TO yyyy***

***mmmm DUPLICATE NODES FOUND***



If no duplicated nodes are found, the following messages will be printed in the message prompt window and in the DYNIFORM.msg message file located in the work directory:

**CHECKING nnnn NODES FOR DUPLICATE NODES**

**NO DUPLICATE NODE FOUND**

#### 5.4.11 CHECK COINCIDENT NODES



This option allows the user to check and merge coincident nodes in the database. Coincident nodes are defined nodes having a distance between each other that is less than or equal to a specified tolerance. eta/DYNIFORM merges all coincident nodes into the one with the lowest node ID number and sets all non-associated nodes as free nodes.

- Select an option in the displayed dialog box.
- Type the tolerance in the displayed dialog box. eta/DYNIFORM prints the number of coincident nodes in the message prompt window and displays the eta/DYNIFORM QUESTION dialog box asking the user to confirm the merge node operation.
- Select YES to merge the coincident nodes.

#### 5.4.12 PART CONNECT



This function displays the names and part identification numbers (PID) of all of the parts that share the selected node.

#### 5.4.13 COMPACT NODE



This function allows the user to remove either all or selected un-referenced nodes in the database and renumbers the remaining nodes.

- Select nodes. Refer to Section 5.4.4.
- Key in a starting node number.

#### 5.4.14 CHANGE NODE NUMBER



This function allows the user to modify the node ID numbers.

1. **NODE**  
Select a node by cursor, or select the KEY IN option (step 2).
2. **KEY IN NODE NO.**  
Key in the existing node number and new node number in the displayed dialog box.
3. After a node is selected, eta/DYNAFORM prints the following message:

***ENTER NEW NODE NO FOR xxx***

If the input node number exists in the database, a message will be printed in the message prompt window:

***NODE NUMBER xxx ALREADY EXISTS, REQUEST DENIED***

***ENTER NEW NODE NUMBER FOR xxx***

4. **UNDO LAST**  
Reject the last operation.

#### 5.4.15 RENUMBER NODES



This function is used to renumber all nodes in the database. There is also an option for creating a summary table of the node ranges for each part. The user may renumber nodes in sequence or by parts.

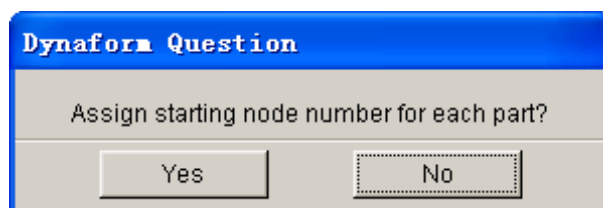


Figure 5.4.4 Starting Node Number Question dialog box

- A eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.4.4 is displayed.
  1. Select YES  
eta/DYNAFORM prints the following message in the message prompt window:

***ENTER STARTING NODE NO. FOR PART? xxxx***

Once a node number is entered, the following message is printed:

***NODE x TO xx ASSIGNED, NEXT NODE NO.: xxxx***

***ENTER STARTING NODE NUMBER FOR PART? xxxx***

The renumbering operation will continue for every part in the database. If the user key in an existing starting node number, eta/DYNAFORM prints the message below, and bring the user to the beginning of this step.

***NODE NUMBER HAS BEEN ASSIGNED***

2. Select NO  
eta/DYNAFORM prints the following message in the message prompt window:

***ENTER STARTING NODE NUMBER***

Once a number has been entered, the following message is printed:

***ENTER NODE NUMBER INC. BETWEEN PARTS***

***0 - DEFAULT TO 1000, -1 - NO GAP BETWEEN PARTS***

#### 5.4.16 DISTANCE BETWEEN NODES/POINTS



This function allows the user to measure the distance between two nodes, two points or a combination of both. The nodes/points are selected by using options provided in the INPUT COORDINATE dialog box. The distance is measured automatically and printed in the message prompt window.

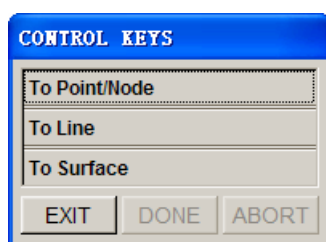


Figure 5.4.5 Measure Distance dialog box

#### 5.4.17 IDENTIFY NODE/POINT



This function allows the user to identify a node/point with its ID number and X, Y, Z coordinates in the global system printed in the message prompt window. Select a node or a point by cursor.

#### 5.4.18 FIND NODE



This function allows the user to find the location and coordinates of a node specified by the input node ID number.

### 5.5 MODEL CHECK & REPAIR (CTRL + R)

This menu provides a convenient way of repairing mesh. It contains the most frequently used functions from the NODE, ELEMENT and MODEL CHECK menus aiding to ease the mesh repair operation. Figure 5.5.1 shows all the functions in the MESH REPAIR dialog box. These functions are: CREATE ELEMENT, MODIFY ELEMENT, DELETE ELEMENT, SPLIT ELEMENT, MOVE NODE, COINCIDENT NODE, NODE BETWEEN POINTS, PROJECT NODES, SHOW BOUNDARY, AUTO NORMAL, ELEMENT SIZE, GAP REPAIR AND SHRINK and LABEL ELEMENT.

A detailed description of these functions is included in their respective sections.

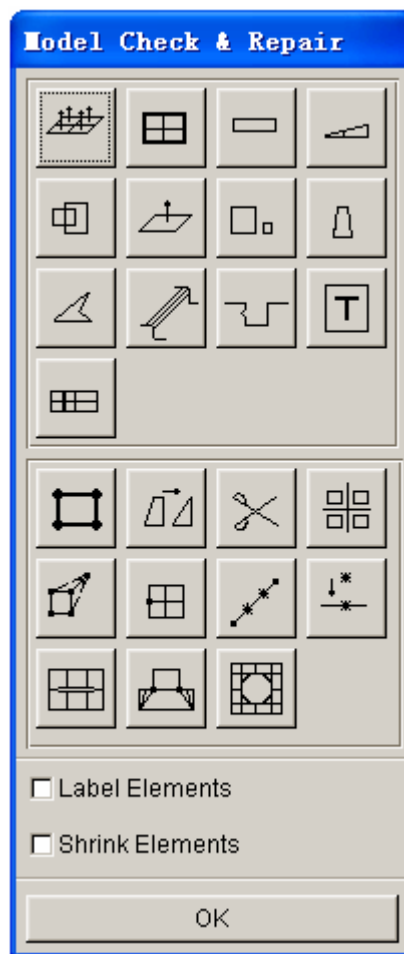


Figure 5.5.1 Model Check &amp; Repair dialog box

### 5.5.1 MODEL CHECK

The functions in the MODEL CHECK menu are used to check the model to meet a specific forming simulation criteria. The functions in this menu are illustrated in Figure 5.5.2.

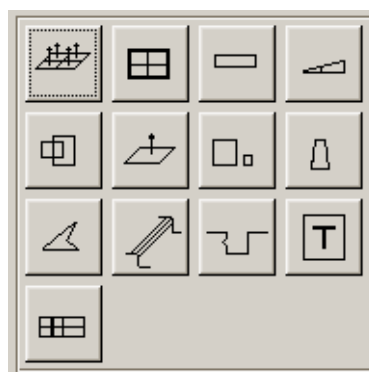


Figure 5.5.2 Model Check

A detailed description of these functions is given in the following sections.

### 5.5.1.1 AUTO PLATE NORMAL

This function allows the user to make consistent normal direction for all elements of the selected part(s).

- Two options are used to select elements: ALL CURRENT PARTS and CURSOR PICK PART. eta/DYNAFORM will display the normal direction of the selected element.
- A eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.5.3 is displayed.

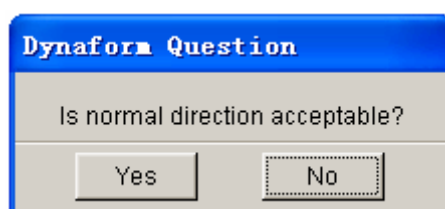


Figure 5.5.3 DYNAFORM Question dialog box

Select YES to accept the normal direction of all elements.

Select NO to reverse the normal direction of all elements.

***NOTE: The user is recommended to select one part at a time. Parts with branches may not have consistent normal direction.***

### 5.5.1.2 BOUNDARY DISPLAY

This function allows the user to check the boundary of displayed part(s). It is used to check the connectivity of elements.

eta/DYNAFORM highlights the free edges along a single surface. For example, if three plate elements share a common edge, the common edge will be displayed as a boundary. An illustration is shown in Figure 5.5.4.

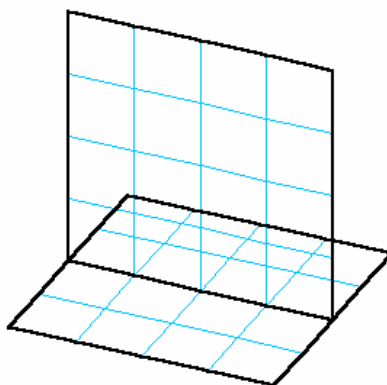


Figure 5.5.4 Surface Boundaries

**NOTE:** *The boundary will remain highlighted until the user hits the CLEAR icon from the ICON bar.*

### 5.5.1.3 ASPECT RATIO CHECK



This function allows the user to check the aspect ratio of the displayed plate and solid elements. The aspect ratio is the ratio of the longest side to the shortest side of an element. In eta/DYNAFORM, the default aspect ratio is 8.0 to 1.0. The user may adjust the aspect ratio accordingly.

1. The user may accept the default value of 8.0 or enter any positive real number. eta/DYNAFORM checks the elements with respect to aspect ratio. Elements that exceed the defined aspect ratio are highlighted. The following message will be printed in the message prompt window:

***xxx ELEMENTS FAILED CHECK ASPECT RATIO= xxx***

or

***ALL ELEMENTS PASS CHECK***

The number of failed elements and largest aspect ratio will be printed in the message prompt window.

2. A eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.5.5 is displayed.

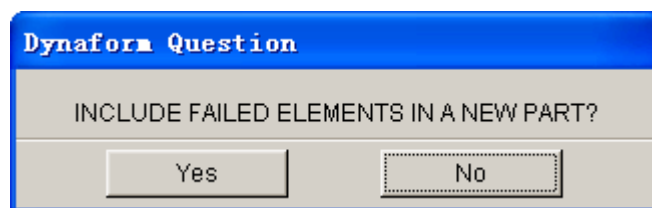


Figure 5.5.5 DYNAFORM Question dialog box

Select YES to display the CREATE PART dialog box.

Select NO to skip to the next function.

3. eta/DYNAFORM prints the following message:

***SELECT ELEMENT FOR ASPECT RATIO***

The user can select any highlighted element to list its aspect ratio in the message prompt window.

#### 5.5.1.4 INTERIOR ANGLE CHECK



This function allows the user to check the minimum values of the interior angles for shell and solid elements. eta/DYNAFORM checks and highlights any elements with interior angles that are less than the defined criteria. The user has an option to adjust the default criteria accordingly.

1. Enter the interior angle in the displayed dialog box (DEFAULT = 1°). If the elements pass the interior angle check, eta/DYNAFORM prints the following message and goes back to the MODEL CHECK menu:

***ALL ELEMENTS PASS CHECK!***

If some elements does not pass the check, eta/DYNAFORM prints the following message with an option to create a new part.

***xxx ELEMENTS FAILED CHECK***

2. At same time, a eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.5.6 is displayed. If ADD TO NEW PART is selected, the CREATE PART dialog box is



displayed to enable generation of new part. If DELETE is selected, all failed elements are deleted. The CANCEL option dismisses the operation.

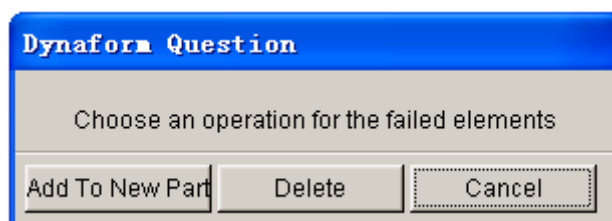


Figure 5.5.6 DYNAFORM Question dialog box

#### 5.5.1.5 OVERLAP ELEMENT



This function allows the user to check overlapping elements that share two or more edges. If there are overlapping elements, the user can include these elements in a new part.

#### 5.5.1.6 PLATE NORMAL



This function is used to draw a boundary line between elements with opposite normal directions. If there are no opposing plate normals, eta/DYNAFORM prints the following message in message prompt window:

***NORMAL CHECK COMPLETED, NORMAL IS CONSISTENT***

#### 5.5.1.7 ELEMENT SIZE



This function is used to check the minimum edge length of plate, solid, or beam elements. eta/DYNAFORM highlights all elements which does not pass the user-defined length criteria. The default is 1.0. The user may change the default value.

- The user can include the failed elements in a new part.
- The user can delete all failed elements.

### 5.5.1.8 CHECK TAPER



This function allows the user to check the taper for quadrilateral elements. Taper value is an amount of convergence. A perfect element such as a rectangle has a taper value of zero. The user can enter a value between 0 and 1 in the displayed dialog box (default is 0.5).

- The user can include the failed elements in a new part.
- The user can select any element to list its taper value in the message prompt window.

### 5.5.1.9 CHECK WARPAGE



This function allows the user to check the warpage angle of quadrilateral elements. The user has an option to split the quadrilateral elements into two triangular elements. All elements that fail the defined criteria may be added to a new part.

1. Enter the warpage criteria (default is 5.0°).
2. If elements do not pass the criteria, a eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.5.7 is displayed.

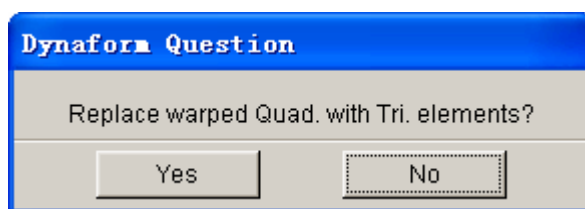


Figure 5.5.7 DYNAFORM Question dialog box

Select YES to replace warped quadrilaterals element with triangular elements.

Select NO to reject the operation.

3. If the warped elements are not replaced, eta/DYNAFORM will ask the user to include the failed elements in the new part or continue to the next step.
4. The user can select any element to list its warpage angle in the message prompt window.

#### 5.5.1.10 FEATURE LINE



This function allows the user to check the angle between plate elements normal with user defined angle. Feature line is the edge of two adjacent plate elements with the angle between the plate normal equal or greater than the angle criteria.

1. Enter angle criteria (default is 20°).
2. Any feature line where the angle between the normal of adjacent elements is larger or equal to the angle criteria will be highlighted. eta/DYNAFORM provides options for user to generate the plotel elements.
3. If YES is selected, eta/DYNAFORM provides options for user to keep the element in a new part. The user creates new part using the CREATE PART dialog box.

#### 5.5.1.11 DIE LOCK



This function is used to check if the die surface has any back draft wall angle. The program checks for an undercut of the elements of the die surface based on default angle criteria. The user is given an option to keep the failed elements in a new part.

#### 5.5.1.12 TIME STEP



This function will display 10 blank elements that have the minimum time step, i.e., the 10 smallest blank elements. Both the time step and element ID number are printed in the message prompt window.

#### 5.5.1.13 SECTION CUT



There are several functions provided in the SECTION CUT dialog box as illustrated in Figure 5.5.8.

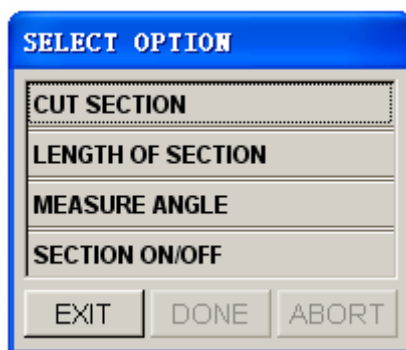


Figure 5.5.8 Section Cut dialog box

### 1. CUT SECTION

This function allows the user to generate section lines that are intersections of the displayed finite element mesh with a set of parallel planes.

- Define a coordinate system by using the options provided in the LCS dialog box.
- Enter the starting location in W-COORDINATE for the first plane in the window, SECTION LOCATION, or click the STARTING SECTION button to select a node or point for the location.
- Enter the end location in W-COORDINATE for the last plane that is perpendicular to the W-axis, or click the END LOCATION button to select a node or point for the location.
- Enter the number of section planes along the W-axis in the window, NUMBER OF SECTIONS.
- eta/DYNAFORM will create a new part named SECTION which stores all section lines.

### 2. LENGTH OF SECTION

This function allows the user to measure the arc length of a line.

There are three options available in this function:

- **CURSOR LOCATION**  
Select two points on a line. eta/DYNAFORM measures the arc length between two points and prints the following message in message prompt window:

***LINE SEGMENT LENGTH IS: xxx***

- **LINE**  
Select the desired line by cursor. The entire length of this section line will be measured and the length is printed in the message prompt window.
  - **REJECT LAST**  
Reject the last selection.
3. **MEASURE ANGLE**  
This function allows the user to measure the angle between two segments on a line.
- Select the desired line by cursor.
  - Select two points on the line to define the linear regression line 1 in red. The user can reset the selection by using REJECT LAST POINT.
  - Select another two points on the line to define the regression line 2 in red. The user can reset the selection by using REJECT LAST POINT.
  - eta/DYNAFORM measures the angle between the two regression lines. The result is printed in the message prompt window.
4. **SECTION ON/OFF**  
This function hides parts except for the part containing section lines. If toggled again, all hidden parts are displayed.

### 5.5.2 MODEL REPAIR

The Model Repair integrates many functions in Node, Element and other Preprocess menus. There are some functions such as Create Element, Modify Element, Delete Element, Split Element, Move Node, Coincident Node, Node between Points, Project Nodes, Gap Repair, Auto Repair and Auto fill. The user can refer to the corresponding sections about those functions for more detailed information.

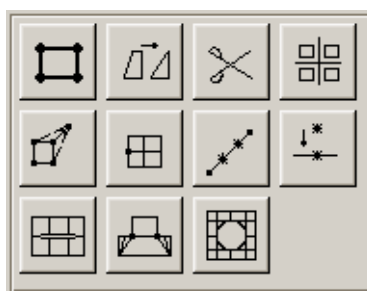


Figure 5.5.9 Model Repair

## 5.6 BOUNDARY CONDITIONS (CTRL+U)

The functions provided in the BOUNDARY CONDITIONS dialog box, as illustrated in Figure 5.6.1, allow the user to create and verify constraints and loads on finite element models.

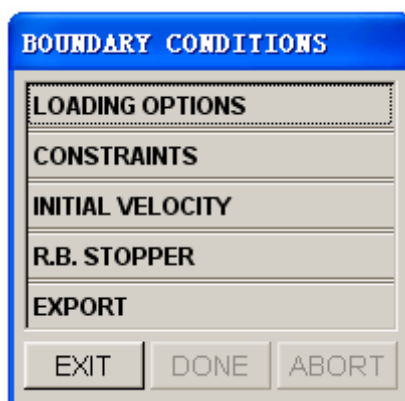


Figure 5.6.1 Boundary Conditions dialog box

### 5.6.1 LOADING OPTIONS

The LOAD SET dialog box is illustrated in Figure 5.6.2.

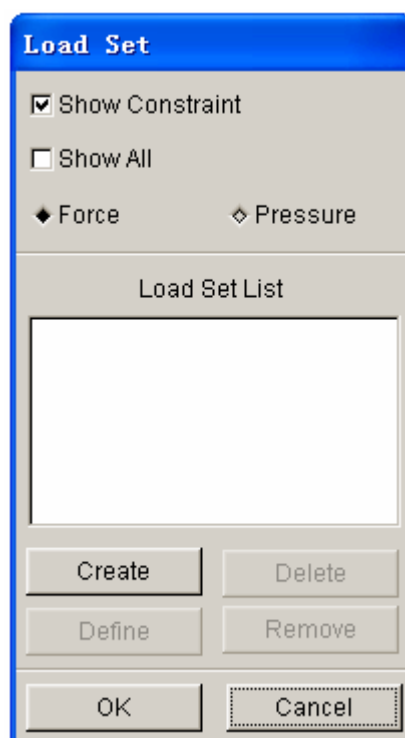


Figure 5.6.2 Load Set dialog box

### SHOW CONSTRAINT

Toggle the checkbox on/off to show/hide **CONSTRAINT**. Refer to **CONSTRAINTS OPTIONS**.

### SHOW ALL

Toggle on/off to show/hide all **LOADING** constraints.

1. Create **FORCE** to be applied on selected nodes.
  - Toggle **FORCE** on.
  - Select **Create**.
  - Enter the load set number in the displayed dialog box.
  - Key in **DOF**, **LOAD CURVE ID** and **SCALE FACTOR** in the **NODE FORCE** dialog box as illustrated in Figure 5.6.3.

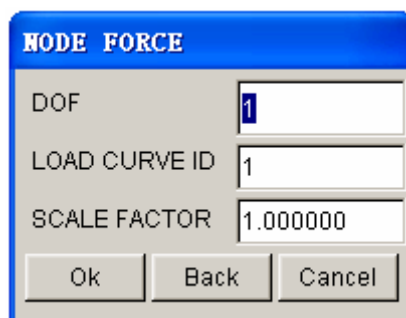


Figure 5.6.3 Define Node Force

### DOF

Degree of freedom. The six DOF is listed as the following:

- 1 - X (X-translation will be loaded.)
- 2 - Y (Y-translation will be loaded.)
- 3 - Z (Z-translation will be loaded.)
- 4 - RX (X-rotation will be loaded.)
- 5 - RY (Y-rotation will be loaded.)
- 6 - RZ (Z-rotation will be loaded.)

### LOAD CURVE ID

Load curve describes amount of load versus time. Refer to Section 12.12.1, **UTILITIES/LOAD CURVE/CREATE CURVE** for information about load curve definition.

### SCALE FACTOR

Scale factor to amplify or reduce amount of load.

- Select OK as illustrated in Figure 5.7.3 to select nodes. Refer to Figure 5.4.2 for information about selecting nodes.
  - Select CANCEL in Figure 5.7.3 to exit.
2. Create PRESSURE to be applied on selected elements.
- Toggle PRESSURE on.
  - Select Create.
  - Enter the load set number in the displayed dialog box.
  - Define input fields in ELEMENT PRESSURE dialog box as illustrated in Figure 5.6.4.

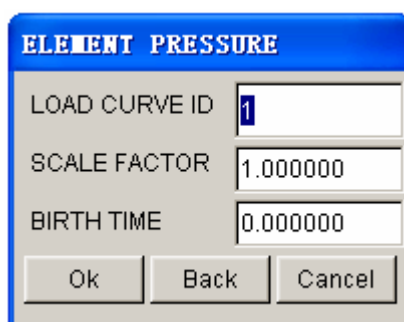


Figure 5.6.4 Element Pressure dialog box

### LOAD CURVE ID

Refer to Create FORCE described above.

### SCALE FACTOR

Refer to Create FORCE described about.

### BIRTH TIME

Time to impose motion or loading.



- Select OK in Figure 5.6.4 to select the element to load.
- Select CANCEL in Figure 5.6.4 to exit.

The following three functions are used for both applying nodal force and element pressure.

### **DELETE**

A eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.6.5 is displayed.

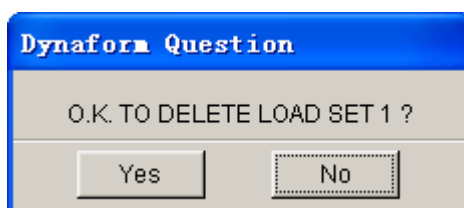


Figure 5.6.5 DYNAFORM Question dialog box

Select YES to delete the last selected load set.

Select NO to reject the choice.

### **DEFINE**

Includes more nodes or elements to a selected load set.

### **REMOVE**

Removes some selected nodes/elements from a load set.

## **5.6.2 CONSTRAINTS**

This function allows the user to create and/or modify the constraints. The CONTSTRAINTS dialog box is illustrated in Figure 5.6.6.

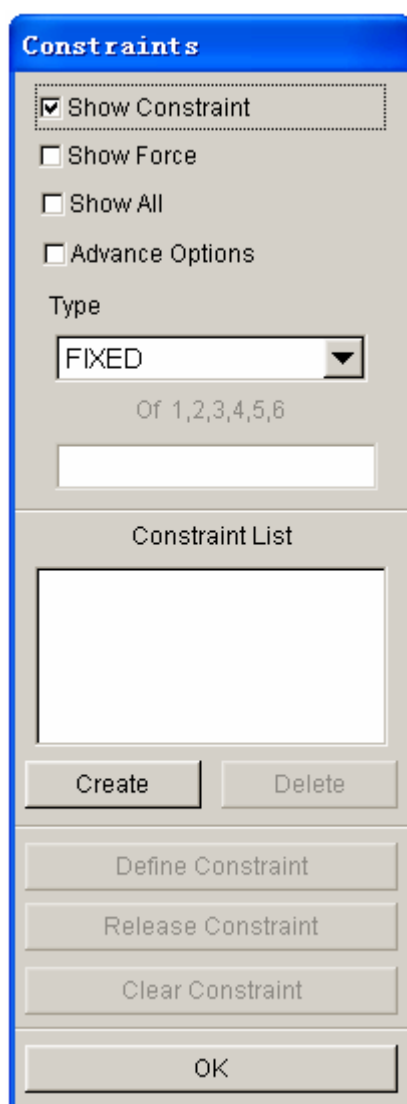


Figure 5.6.6 Constraints dialog box

### **SHOW CONSTRAINT**

Toggle checkbox on/off to show/hide display of selected CONSTRAINT set.

### **SHOW FORCE**

Toggle checkbox on/off to show/hide load force.

### **SHOW ALL**

Toggle checkbox on/off to show/hide all Constraints.

## ADVANCED OPTIONS

Toggle checkbox on to access the advanced options for constraint definition.

### 1. CREATE CONSTRAINT SET

- Select a type of constraint. In LS-DYNA, each node is associated with six degree of freedom as listed below:

1. X-Translation
2. Y-Translation
3. Z-Translation
4. X-Rotation
5. Y-Rotation
6. Z-Rotation

The following drop down menu as illustrated in Figure 5.6.7 provides the basic and advanced approaches for definition of constraints.

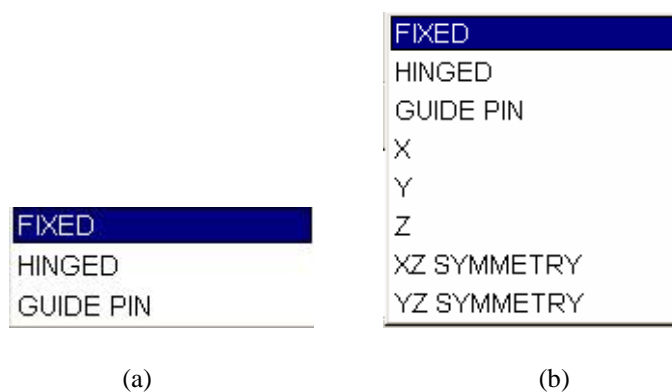


Figure 5.6.7 Type of Constraint (a) Basic (b) Advanced

- Select CREATE.
  - Enter the Constraint Set ID Number in the displayed dialog box.
  - Select nodes to apply constraints.
- ### 2. DELETE CONSTRAINT SET
- Select a constraint set from the list.
  - Select DELETE.

- Select Yes to remove the constraint set from the eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.6.8.

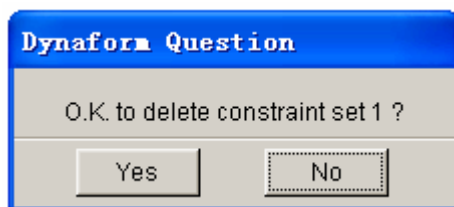


Figure 5.6.8 DYNAFORM Question dialog box

### 3. DEFINE CONSTRAINT

- Select a set from the list.
- Select type of constraint.
- Select DEFINE.
- Select nodes. Nodes with the selected degree of freedom will be added to the highlighted constraint set.

### 4. RELEASE CONSTRAINT

Remove some nodes from the selected constraint set.

### 5. CLEAR CONSTRAINT

Remove all nodes from a selected constraint set. A eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.6.9 is displayed.

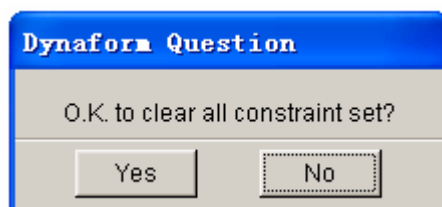


Figure 5.6.9 DYNAFORM Question dialog box

Select YES to remove all constraints.

Select NO to reject the choice.

### 5.6.3 INITIAL VELOCITY

This function allows the user to define and apply linear or angular velocities to selected nodes. The VELOCITY SET dialog box is illustrated in Figure 5.6.10.

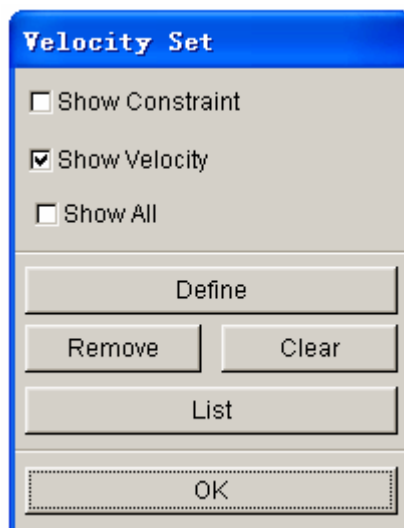


Figure 5.6.10 Velocity Set dialog box

#### SHOW CONSTRAINT

Refer to CONSTRAINT OPTION.

#### SHOW VELOCITY

Toggle on/off to show/hide a defined initial velocity.

#### SHOW ALL

Toggle on/off to show/hide all initial velocity.

#### DEFINE

Create a velocity set.

- Select DEFINE.
- Enter the initial velocity in the INTIAL VELOCITY dialog box as illustrated in Figure 5.6.11.
- Select nodes.

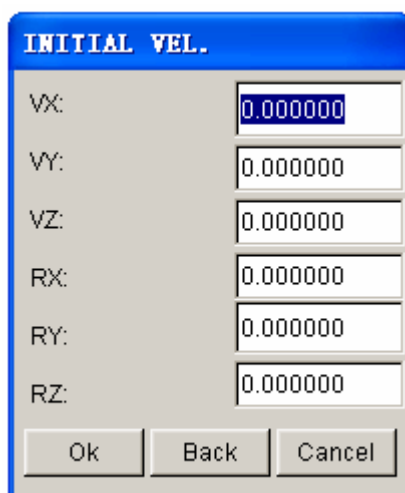


Figure 5.6.11 Initial Velocity dialog box

### REMOVE

Remove defined initial velocity from the selected nodes set.

### CLEAR

Remove all defined initial velocity from the database. A eta/DYNAFORM QUESTION dialog box as illustrated in Figure 5.6.12 is displayed.

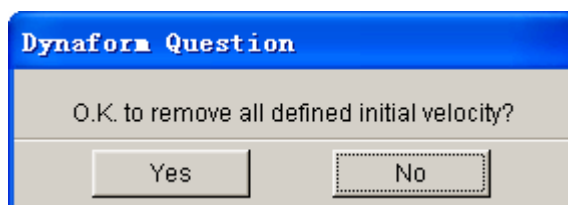


Figure 5.6.12 DYNAFORM Question dialog box

### LIST

There are two choices:

1. ALL NODE  
All nodes with an initial velocity are highlighted with an arrow.
2. NODE  
The initial velocities of selected nodes are listed in the message prompt window.

### 5.6.4 RIGID BODY STOPPER

This function allows the user to define a RIGID BODY STOPPER. A RIGID BODY STOPPER provides a convenient way of controlling the motion of rigid tooling during the forming process. For a detailed description, refer to the LS-DYNA User's Manual. The available options are shown in Figure 5.6.13.

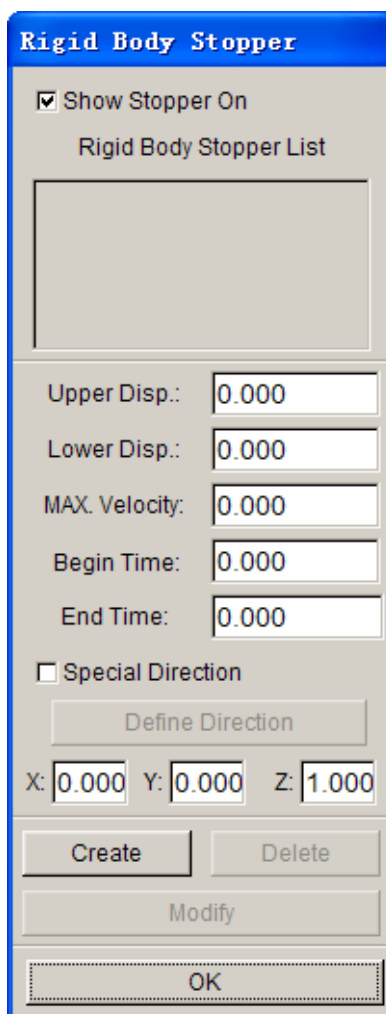


Figure 5.6.13 Rigid Body Stopper dialog box

- Enter the value in the input field to define the distance from the upper stopper (or lower stopper) to the master part.
- Select CREATE. The SELECT PART dialog window is displayed. The user can select a part as the master part.
- Select DELETE to remove the defined rigid body stopper. Select Yes from the eta/DYNAFORM dialog box as illustrated in Figure 5.7.14 to remove the definition of rigid body stopper.



Figure 5.6.14 DYNAFORM Question dialog box

- After changing the UP or DOWN value, select MODIFY. The user can change the position of the RIGID BODY STOPPER.
- Select the SHOW STOPPER ON window to show a stopper in the list.

### 5.6.5 EXPORT

This function allows user to export the defined boundary conditions in eta/DYNAFORM to a text file using LS-DYNA format. After the function is selected, the dialog box as illustrated in Figure 5.6.15 is displayed. The user can select the type of boundary condition which will be exported.

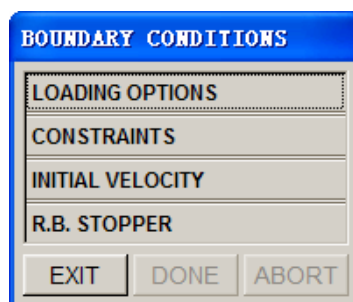


Figure 5.6.15 Boundary Conditions dialog box

### 5.7 NODE/ELEMENT SET (CTRL+V)

This function allows the user to create node and element sets to facilitate the organization of output data. The available options are shown in Figure 5.7.1.



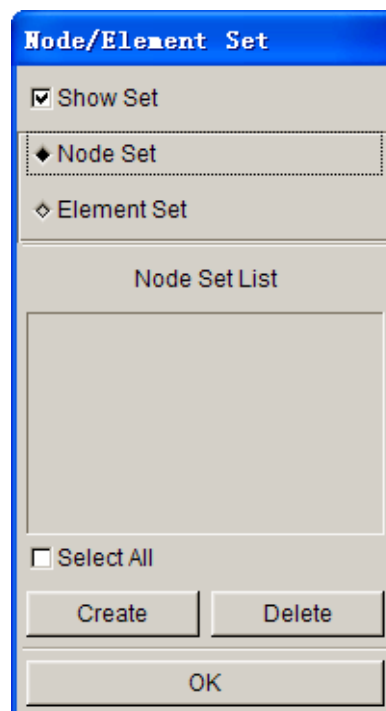


Figure 5.7.1 Node/Element Set dialog box

- Toggle checkbox on/off to show/hide NODE SET or ELEMENT SET.
- Select CREATE.
- Select nodes or elements in the corresponding dialog box.
- Select DELETE to delete a set that is highlighted in the list.

## Chapter 6

### DFE (DIE FACE ENGINEERING)

The eta/DYNAFORM-DFE module, Die Face Engineering, has been developed to generate Die Face (Addendum and Binder) to support early stage die design. The functions provided in the eta/DYNAFORM-DFE module are shown in Figure 6.1.

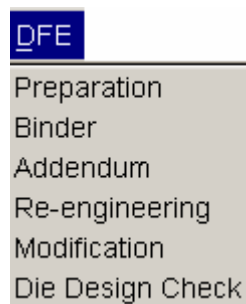


Figure 6.1 DFE Menu

Features of the DFE module include the following highlights.

- Flange Unfolding
- Group surface and create mid surface from a solid
- Automatic filleting for sharp edges
- Automatic filling of holes, boundary fill and outer boundary smooth
- Automatic tipping and adjusting of drawing direction
- Repair the topology of mesh
- Automatic and interactive generation and modification of a binder
- Automatic and interactive generation and modification of outer and inner addendum
- Morphing part or die geometries
- Re-engineering
- Drawbar creation
- Full parameterization

A detailed description of each function and its corresponding submenu is given in the following sections.

## 6.1 PREPARATION

The functions in this menu allow the user to prepare necessary data for the part prior to the die face design. The user may import part geometry or part mesh using DFE/PREPARATION/IMPORT functions. Refer to Section 3.6, IMPORT, for a detailed description of importing geometry file and mesh file.

The available functions in the DFE/PREPARATION menu are shown in Figure 6.1.1.

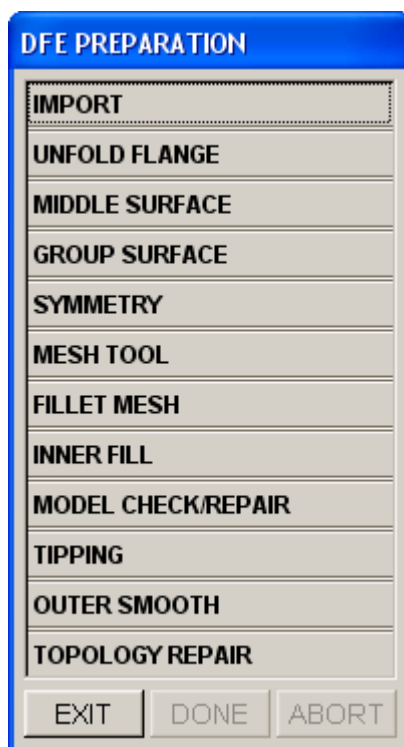


Figure 6.1.1 Preparation Menu

### 6.1.1 IMPORT

This function allows the user to read in CAD or finite element model data into active database. Please refer to Section 3.6 for detailed information about the Import function.

### 6.1.2 UNFOLD FLANGE

This function unfolds flanges on a product so the user can create an addendum with a developed product trim line.

- Once the function is selected, eta/DYNAFORM opens the SELECTED SURFACE dialog box to select flange surfaces to be unfolded. Refer to Figure 6.1.2 for typical surfaces with the

defined flange highlighted.

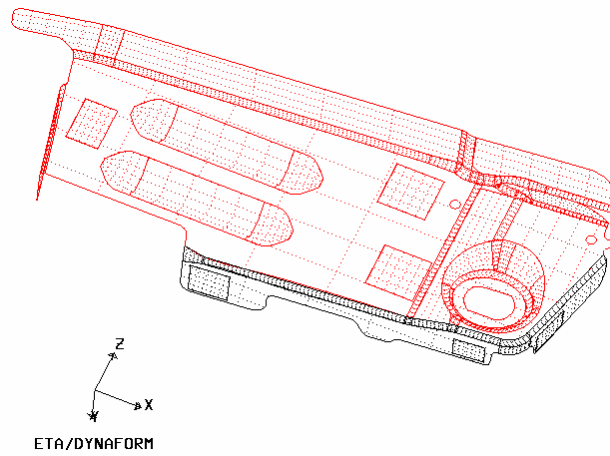


Figure 6.1.2 Select Flange Surface

- After selecting surfaces, the DFE module will determine and highlight the unfold line and the boundary line. As shown in Figure 6.1.3, the bold line is the unfold line, and the other is the boundary line of the base surface and unfolded surface. eta/DYNAFORM displays a SELECT OPTION dialog box as illustrated in Figure 6.1.4. The user has to select an option to continue operation.

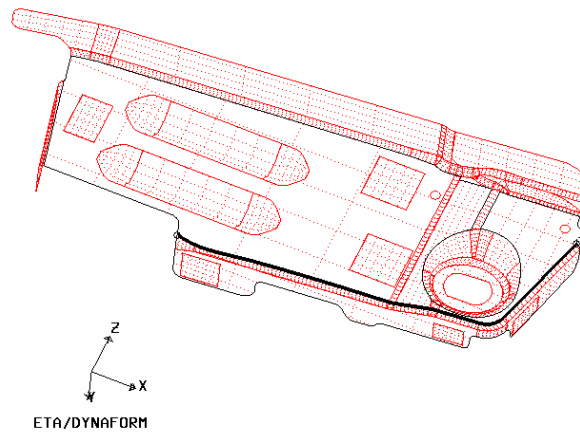


Figure 6.1.3 Unfold Line and Boundary Line

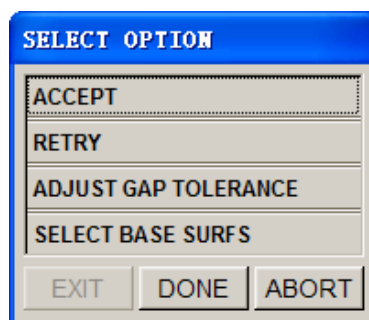


Figure 6.1.4 Select Option dialog box

- If the unfold line and boundary line are correct, select ACCEPT to continue. The program will unfold the selected surfaces along the unfold line and continue to the next step.

If the unfold line is not correct, select RETRY. eta/DYNAFORM will use a different method to find the desired unfold line and boundary line. If ADJUST GAP TOLERANCE is selected, the dialog box as illustrated in Figure 6.1.5 is displayed to allow the user to change the gap tolerance between surfaces.

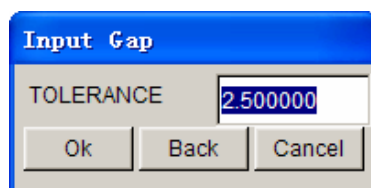


Figure 6.1.5 Input Gap dialog box

The default gap tolerance is 2.5. If there are small surfaces in the part, a smaller gap tolerance should be used. If SELECT BASE SURFS is selected, eta/DYNAFORM allows the user to select surfaces as the base surfaces. A base surface is a surface that connects the flange surfaces along the unfold line.

- The INPUT BENT ANGLE dialog box as illustrated in Figure 6.1.6 is displayed. The bent angle is the angle between the unfolded surface and the base surface. The default angle (0°) indicates that unfolded surfaces are tangent to the base surfaces at the unfold line.

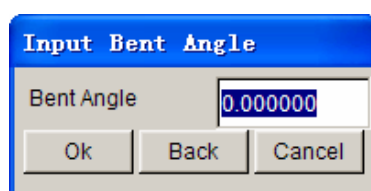


Figure 6.1.6 Input Bent Angle dialog box

- Select OK to accept the defined bent angle. The unfolded surfaces are shown in Figure 6.1.7 in yellow.

The unfolded surfaces are automatically stored in a new part named UNFOLDED.

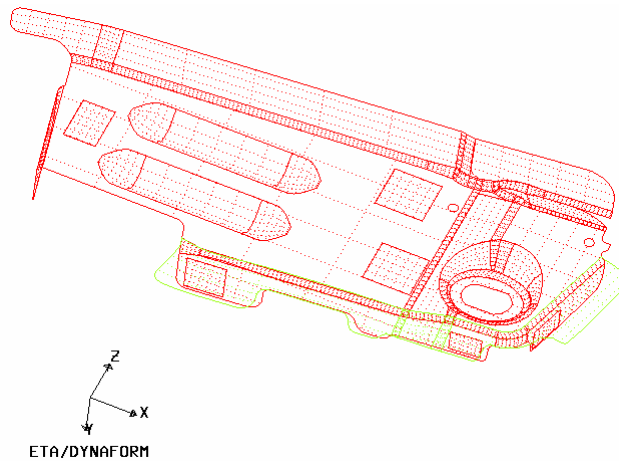


Figure 6.1.7 Unfolded Flange

- Select an option from the CONTROL KEYS dialog box as illustrated in Figure 6.1.8.

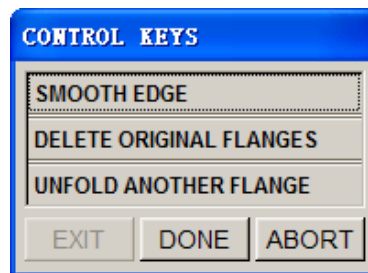


Figure 6.1.8 Control Keys dialog box

### SMOOTH EDGE

This function allows the user to smooth the edge of unfolded surfaces. eta/DYNAFORM will request the user to select the starting and ending points on the edge of the unfolded flange. The smoothed edge will be highlighted as shown in Figure 6.1.9. The CONTROL KEYS dialog box as illustrated in Figure 6.1.10 is displayed. The user has to select the next step from the options provided in the dialog box.

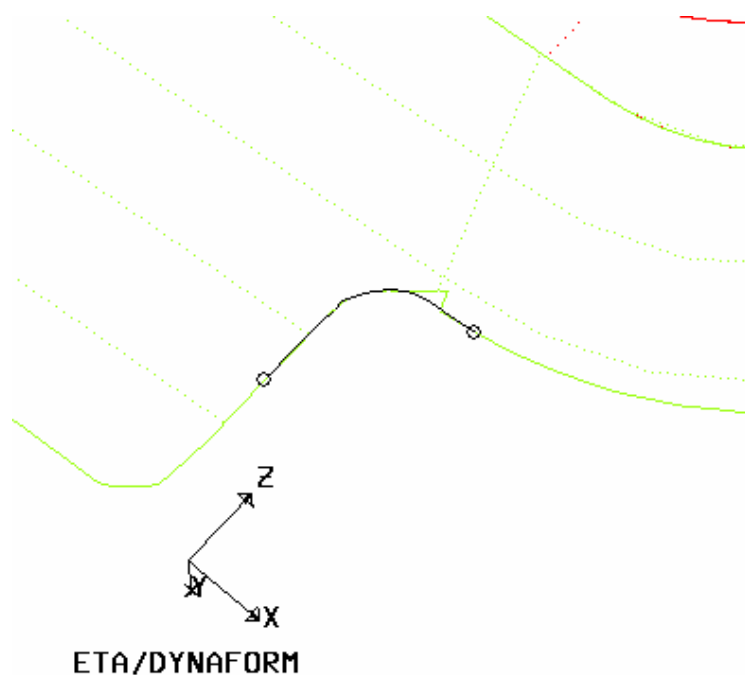


Figure 6.1.9 Smooth Edge



Figure 6.1.10 Control Keys dialog box

**FINISH SMOOTH** – Completes the smooth edge operation.

**SMOOTH AGAIN** – Performs the smooth edge operation again.

**UNDO** – Undoes the last smooth operation.

#### **DELETE ORIGINAL FLANGES**

The original flange surfaces will be removed from the database.

#### **UNFOLD ANOTHER FLANGE**

Continues to unfold another flange by repeating the steps previously mentioned.

### 6.1.3 MIDDLE SURFACE

This function allows user to create the mid surface from a solid part. For more information please refer to the Section 5.2.23.

### 6.1.4 GROUP SURFACE

This function allows user to group the top or the bottom surface from a solid part. For more information please refer to the Section 5.2.24.

### 6.1.5 SYMMETRY

This function allows the user to mirror the symmetrical die about a symmetry plane.

The user has to define a part as DIE prior to defining the symmetry plane. If the DIE is not defined, a eta/DYNAFORM QUESTION dialog box as illustrated in Figure 6.1.11 is displayed when the function is activated.

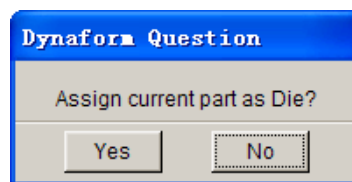


Figure 6.1.11 DYNAFORM Question dialog box

If YES is selected, current part is set as DIE and the user will continue to define the symmetry plane using the functions provided in SYMMETRY dialog box as illustrated in Figure 6.1.12.

If NO is selected, eta/DYNAFORM will abort the operation.



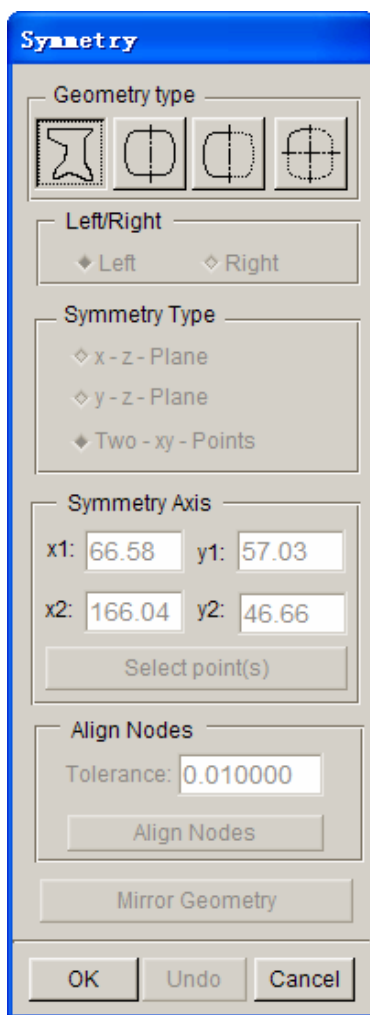


Figure 6.1.12 Symmetry dialog box

Four geometry options are available in the SYMMETRY dialog box. There are described as following:



**NO SYMMETRY (Default)**

This is the default option. All other functions provided in the SYMMETRY dialog box is disabled.



**SYMMETRY**

The part is defined as symmetrical.

**HALF SYMMETRY INPUT**

Only one half of the part geometry is available. The function will mirror the other half of the part about a symmetry plane to create a complete part.

**QUARTER SYMMETRY INPUT**

Only one quarter of the part geometry is available. The function will mirror the other three quarters of the part about two symmetry planes to create a complete part.

There are three types of symmetry options available. They are described as the following:

**X-Z PLANE**

The symmetry plane is parallel to the XZ plane of the Global Coordinate System. The user must select two points to define the XZ plane. The first point is used to define the location of the symmetry plane, while the second point is used to define direction of the axis of symmetry.

**Y-Z PLANE**

The symmetry plane is parallel to the YZ plane of the Global Coordinate System. The user must select two points to define the YZ plane. The first point is used to define the location of the symmetry plane, while the second point is used to define direction of the axis of symmetry.

**TWO-XY-POINT**

The symmetry plane is normal to the XY plane of the Global Coordinate System. The user must select two points to define the XY plane. The first point is used to define the location of the symmetry plane, while the second point is used to define direction of the axis of symmetry.

If the nodes at the plane of symmetry are not collinear along the symmetry plane, the ALIGN NODES function can be utilized to correct the problem. When the part is defined as Half Symmetry or Quarter Symmetry, eta/DYNAFORM will activate the ALIGN NODES function. The user can define the tolerance input field to align the nodes along the line of symmetry. eta/DYNAFORM will show the result from the operation and send the user a message to confirm the operation.

If QUARTER SYMMETRY INPUT is selected, there are two symmetry planes and two symmetry axes. One symmetry axis is vertical to the first and through the base point (first selected point), while the other symmetry axis is vertical to the first plane and through the second symmetry axis.

If HALF SYMMETRY INPUT or QUARTER SYMMETRY is selected, the MIRROR GEOMETRY button will be activated. This function allows the user to select elements which will be mirrored about the symmetry plane(s). The UNDO button is used to undo the mirror operation.

The steps needed to define a symmetrical die are listed as the following:

1. Read in the symmetrical geometry, mesh it and assign it as DIE.
2. Select a geometry type, for example, HALF SYMMETRY.
3. Select a symmetry type, for example, X-Z PLANE.
4. Using the SELECT POINT button to select two points to define the symmetry plane and the axis direction.

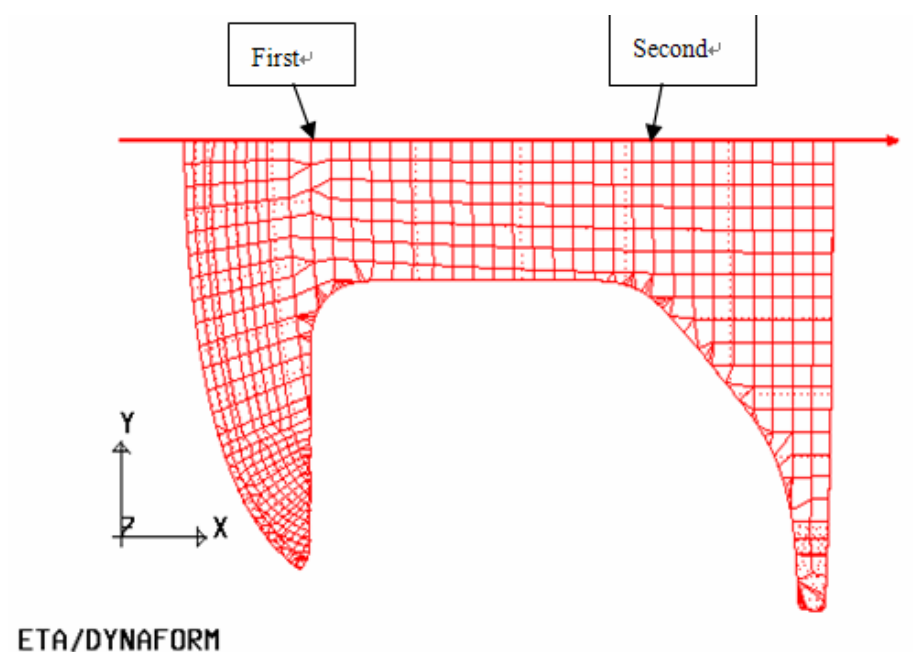


Figure 6.1.13 Half Symmetry Part With X-Z Plane

5. Click on the MIRROR GEOMETRY SYMMETRY button, following by selecting the part as illustrated in Figure 6.1.13. The mirror result is illustrated in Figure 6.1.14.

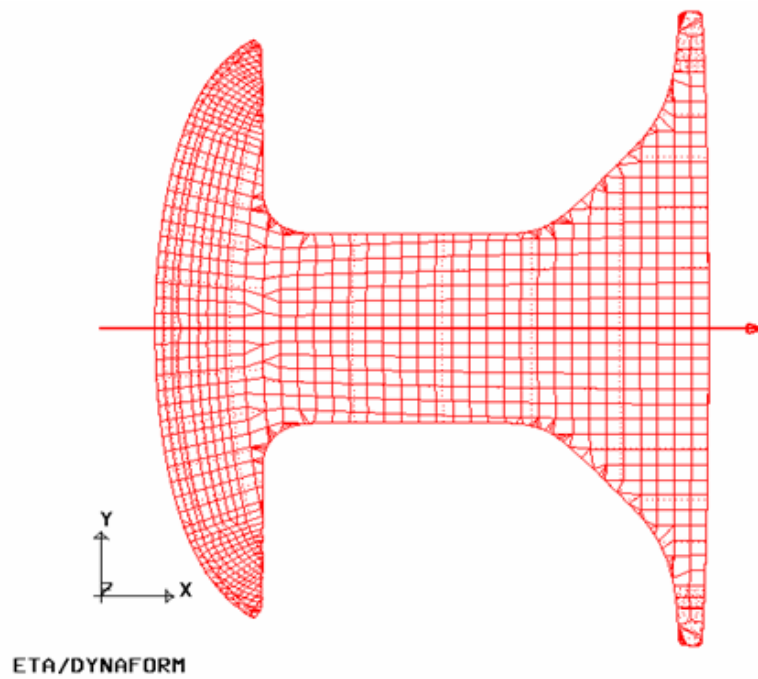


Figure 6.1.14 The Mirroring Result

*NOTE: Once the symmetry axis is defined, the created binder will be symmetrical and consist of two surfaces as shown in Figure 6.1.15.*

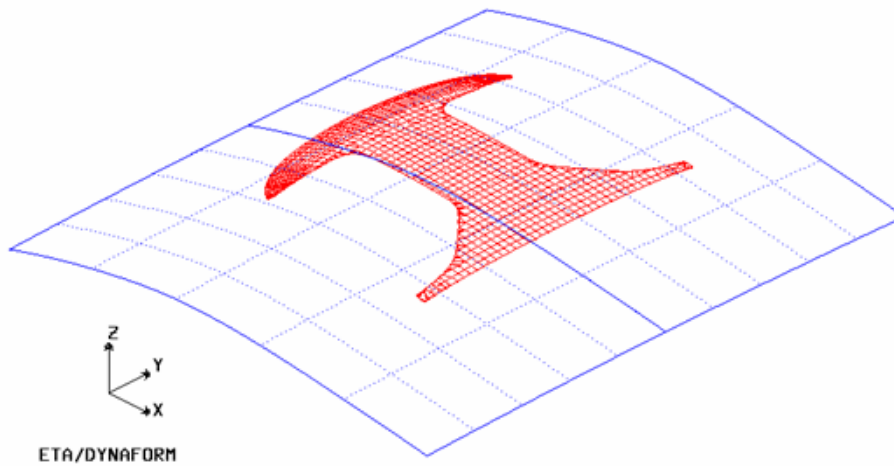


Figure 6.1.15 Binder With Symmetry Plane

### 6.1.6 MESH TOOL

This function is similar to the SURFACE MESH function as described in the Section 5.3.4, SURFACE MESH.

### 6.1.7 FILLET MESH

The FILLET MESH function is used to generate fillet elements at the sharp corners of a part. This function requires a relatively clean mesh without overlap elements or interior gaps, in addition to consistent element normals. The FILLET MESH dialog box as illustrated in Figure 6.1.16, provides options to locate and fillet sharp corners.

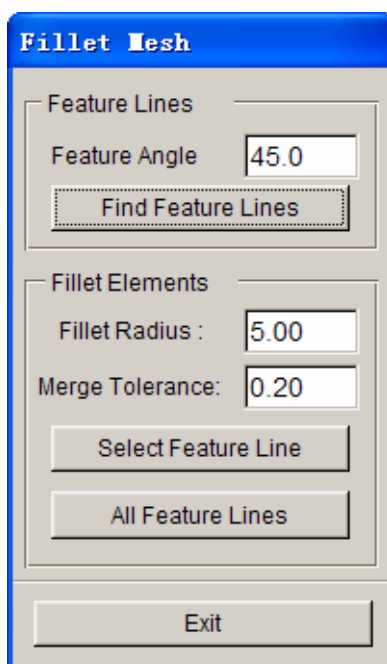


Figure 6.1.16 Element Fillet dialog box

The following steps are required to generate the fillet mesh:

- **FEATURE ANGLE**
  1. Enter the angle criteria (the default is 45°).
  2. Click the FIND FEATURE LINES button to seek the feature lines where the angle is larger than or equal to the angle criteria. The program will highlight all features lines that meet the criteria.
  
- **FILLET ELEMENT**
  1. Key in input field of FILLET RADIUS to define fillet radius for all feature edges.
  2. MERGE TOLERANCE function is used to define the gap tolerance between nodes on the fillet radius. Nodes that are near by each other will be merged if the distance is less than the defined tolerance.
  3. If SELECT FEATURE LINE option is selected, eta/DYNAFORM will display the

SELECT NODE dialog box for the user to select two or three nodes on the feature line. If two nodes are selected on a two-face feature line, eta/DYNAFORM will display a dialog box as shown in Figure 6.1.19. If three nodes on a three-face feature line are selected, eta/DYNAFORM will display a dialog box as illustrated in Figure 6.1.20a.

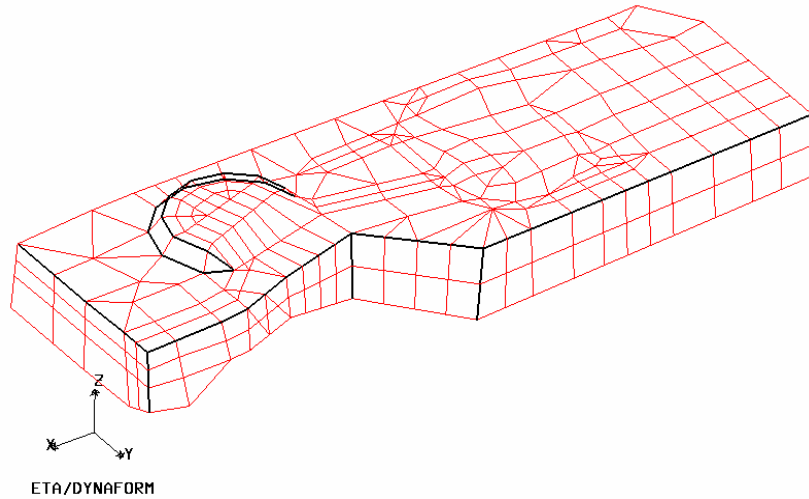


Figure 6.1.17 Find Feature Lines

### ALL FEATURE LINES

When ALL FEATURE LINES function is selected, all feature lines will be filleted with the global radius. Press OK button on the PARAMETERS dialog box to generate the fillet mesh. The FILLET function supports different radii and automatically smooth the elements.

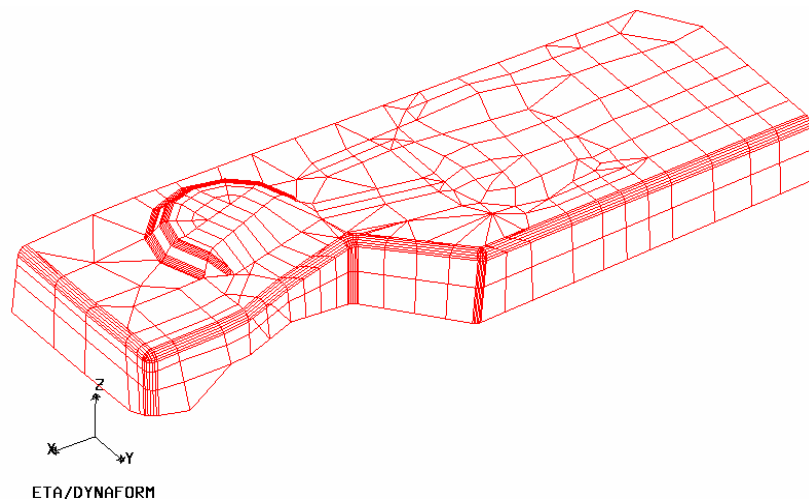


Figure 6.1.18 Fillet with All Feature Lines

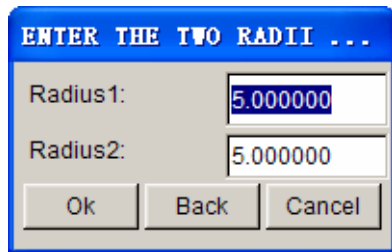
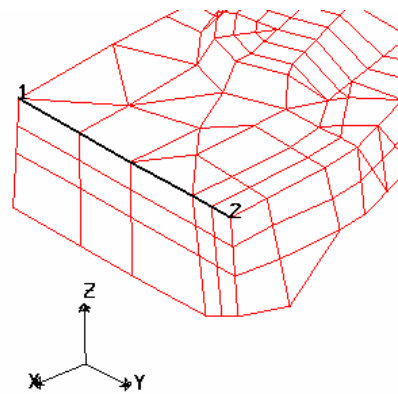
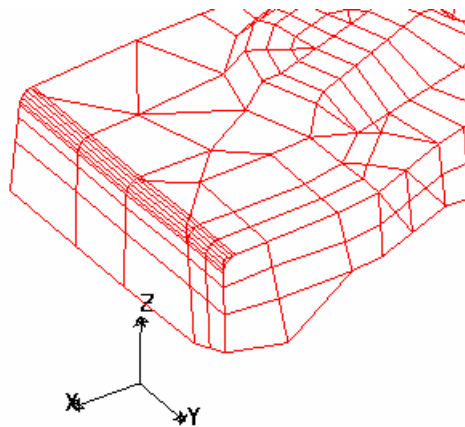


Figure 6.1.19a Element Fillet



ETA/DYNAFORM

Figure 6.1.19b Select feature line



ETA/DYNAFORM

Figure 6.1.19c Element Fillet Result

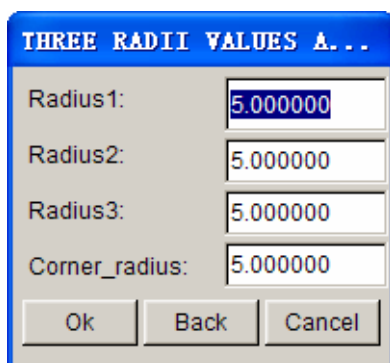
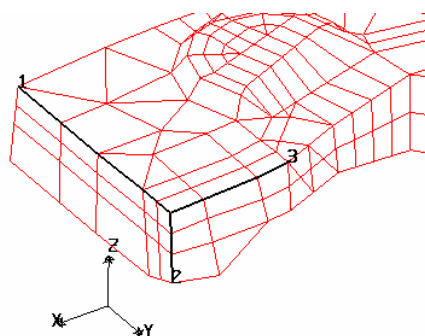


Figure 6.1.20a Element Fillet



ETA/DYNAFORM

Figure 6.1.20b Select Feature Line

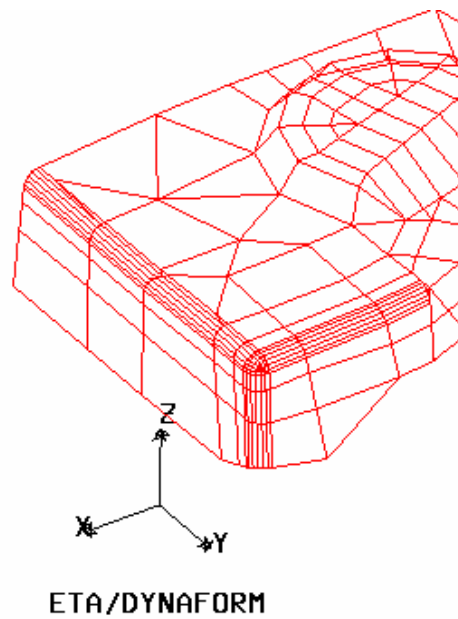


Figure 6.1 20c Element Fillet Result

### 6.1.8 INNER FILL

The INNER FILL function is used to fill holes and missing geometry on outer boundary of the products such as windows, wheel houses, tail light lamps, access holes, bolt holes, etc. There are four functions available in INNER FILL dialog box as illustrated in Figure 6.1.21. These functions are:

1. Auto Fill
2. Hole Fill
3. Curve Fill
4. Polygon Fill



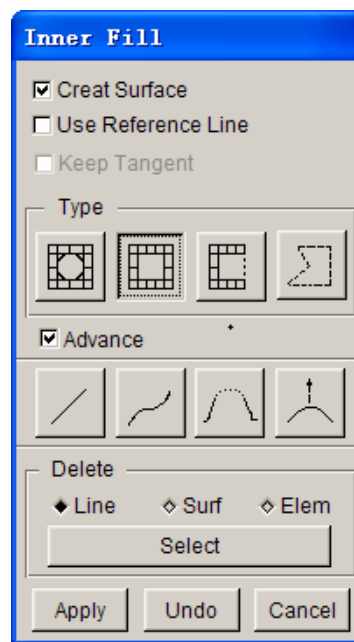


Figure 6.1.21 Inner Fill dialog box

**Create Surface**

This option allows user to decide if surfaces are generated upon completion of the inner fill operation.

**Use Reference Line**

It is an advanced option. For filling complicated surface, the user can create some reference curves to fill the holes.

**Keep Tangent**

This option enables the user to keep tangency between the existed surfaces and surfaces generated by fill operation. Figure 6.1.22 illustrates the surface generated with and without tangency option.

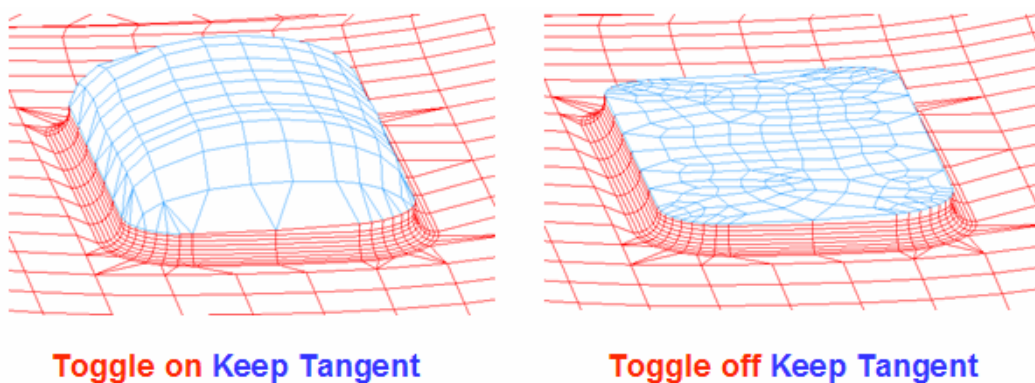


Figure 6.1.22 Toggle on/off Keep Tangent option

**ADVANCED**

The advanced option includes some auxiliary functions such generation and removal of curves, removal of surfaces and elements.

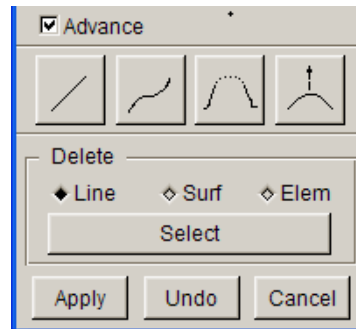






Figure 6.1.23 Advanced Option

-  This function is used to create lines.
-  This function is used to create spline.
-  This function generates a bridge line connecting two selected lines.
-  This function is used to edit the curve.

After clicking the Select button, eta/DYNAFORM will display the Create Line dialog box as illustrated in Figure 6.1.24.

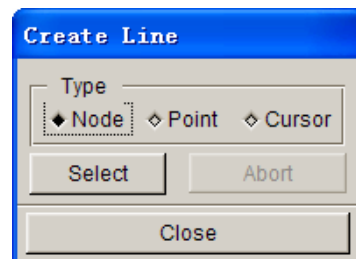


Figure 6.1.24 Create Line dialog box

- **NODE:** Create line through selecting the nodes on display area.
- **POINT:** Create line through selecting the nodes on display area.
- **CURSOR:** Create line through the picking location on display area.

**DELETE**

This option enables the user to remove the selected entities from the database.

### 6.1.8.1 AUTO FILL

This function is used to locate and fill all holes in the part with elements and surfaces. The user may choose whether or not to display the surfaces. This option is controlled in the configuration file: DYNIFORMDEFAULT, located in the installation directory of eta/DYNIFORM.

**NOTE:** Click UNDO on the OPTION dialog box as illustrated in Figure 6.1.21. The filled mesh and surface will be removed.

### 6.1.8.2 HOLE FILL

This function enables user to automatically fill all the holes in the displayed part. After clicking the APPLY button, eta/DYNIFORM highlights all the inner boundary as illustrated in Figure 6.1.25.

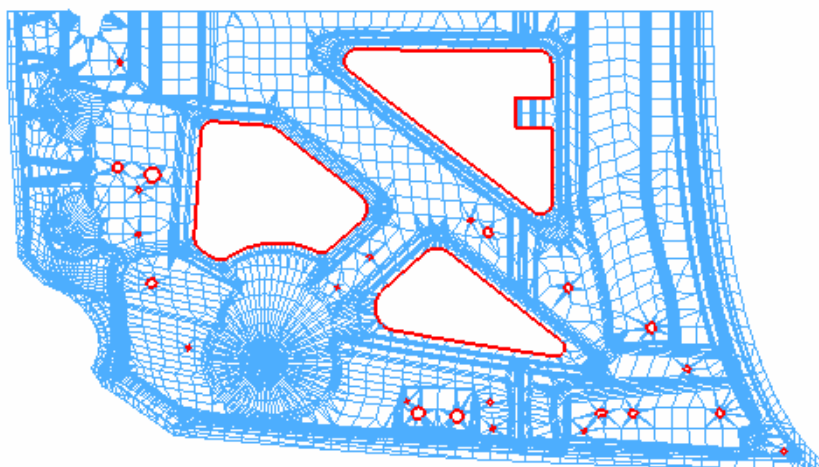


Figure 6.1.25 Boundary Lines of Inner Holes

The user can click on the middle mouse button to fill all the highlight holes. Else, click on the right mouse button to abort the operation. Figure 6.1.26 illustrates an example of inner holes filling operation.

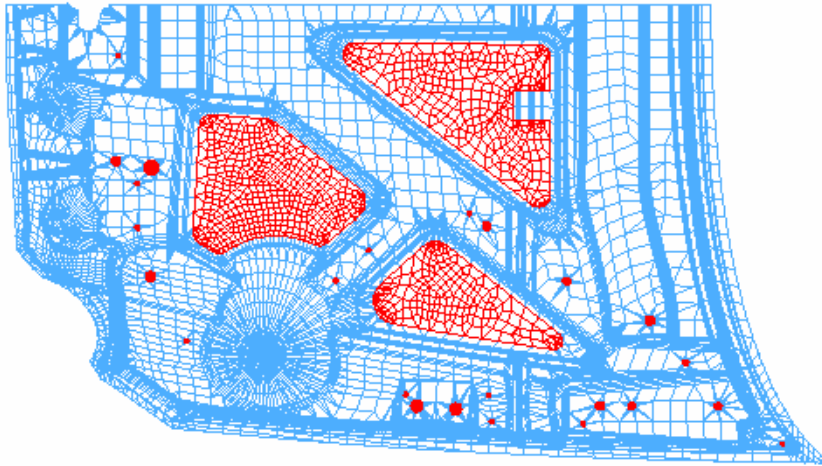


Figure 6.1.26 Filled Mesh within a Closed Curve

If the user wants to exclude some from the operation, he/she can move the mouse cursor around the highlighted holes and click the left mouse button to deselect the holes.

### 6.1.8.3 CURVE FILL

This function allows the user to fill an open or cut-out area. Figure 6.1.27 illustrates an example of Curve Fill operation using a spline curve to close the cut-out area.

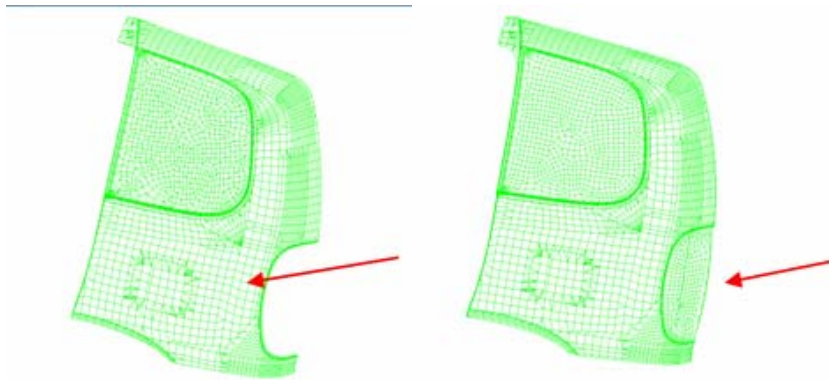


Figure 6.1.27 Filled Mesh with Open Curve

1. If the OPEN CURVE option is selected, eta/DYNAFORM will request the user to select elements neighboring to the cut-out area.
2. Select OK to accept the selected elements and a dialog box as illustrated in Figure 6.1.28 is displayed. The user can select one of the following methods to fill the area:

**SELECT NODES**

This option allows the user to select a set of nodes to define the outer boundary of the open curve.

**SELECT LINE**

This option allows the user to select and define an existing line as the outer boundary of the open curve. After a line is selected, the SELECT NODES dialog box is displayed. The following message is printed in the message prompt window:

***SELECT TWO NODES TO SPLIT THE LINE, OR PICK OK TO USE THE ENTIRE LINE***

After selecting two nodes, the line segment between the selected nodes will be used as the boundary to close the open curve.

**CREATE LINE**

This option allows the user to create a line as the outer boundary to close the open curve for filling. After a line is created, the SELECT NODES dialog box is displayed. The following message is printed in the message prompt window:

***SELECT TWO NODES TO SPLIT THE LINE, OR PICK OK TO USE THE ENTIRE LINE***

After selecting two nodes, the line segment between the selected nodes will be used to close the open curve.

**CREATE SPLINE**

Allow the user to create a spline curve as the outer boundary for fill mesh. After a spline is created, the SELECT NODES dialog box is displayed. The following message is printed in the message prompt window:

***SELECT TWO NODES TO SPLIT THE LINE, OR PICK OK TO USE THE ENTIRE LINE***

After selecting two nodes, the spline segment between the two points will be used.

3. After the curve is defined using any of the above methods, eta/DYNAFORM will fill a set of elements within the region.

#### 6.1.8.4 POLYGON FILL

This function allows the user to select a set of nodes as the boundary edge to create a mesh to fill the boundary. After function is selected, the SELECT NODES dialog box is displayed. The following message is printed in the message prompt window:

***SELECT A SET OF NODES CONSECUTIVELY AS BOUNDARY EDGE***

After a set of nodes is selected, eta/DYNAFORM will generate a set of elements to fill the inner boundary.

#### 6.1.9 MODEL CHECK / MESH REPAIR

All functions in the MODEL CHECK & MESH REPAIR dialog box are described in Sections 5.5. Please refer to the section for a detailed description of these functions.

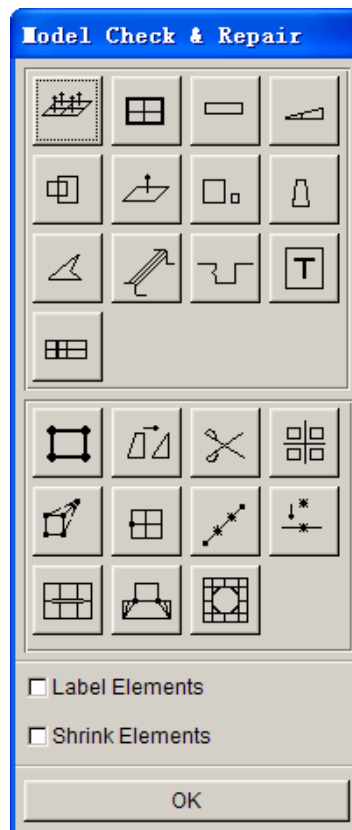


Figure 6.1.28 Model Check dialog box

### 6.1.10 TIPPING

In most stamping simulations, the global Z direction is selected as the stamping direction. Some product design might not be compatible with the default setting of eta/DYNAFORM, therefore the tipping function is used to adjust the product to follow the stamping direction. It allows the user to translate and/or rotate the part about a defined axis. The TIPPING dialog box is illustrated in Figure 6.1.29.

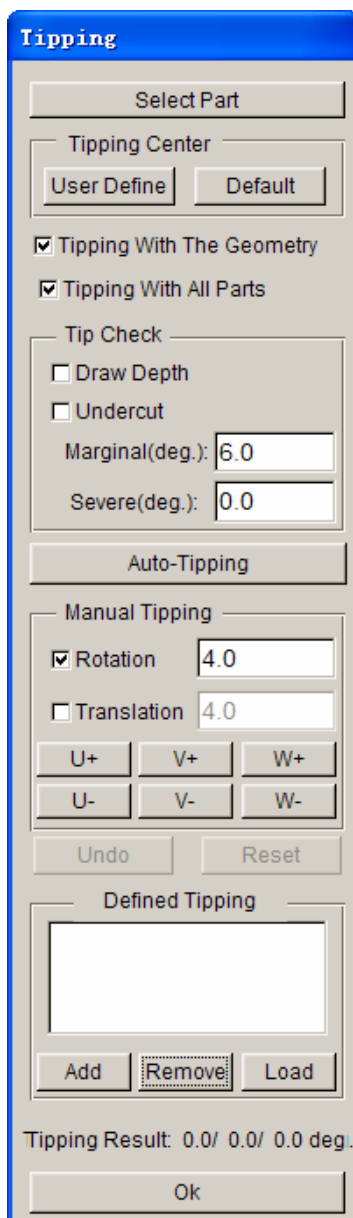


Figure 6.1.29 Tipping dialog box

Part(s) must be defined prior to performing tipping operation. The user can click on the SELECT PART button on the top of the menu to select part(s). All the selected part(s) will be locked as the tipping object.

### 6.1.10.1 TIPPING CENTER

This function is used to define location of tipping center.

- DEFAULT - The default tipping center is set at center of mass.
- USER DEFINE - The user selects a node, point or center coordinate as the tipping center using the INPUT COORDINATE dialog box as illustrated in Figure 6.1.30.

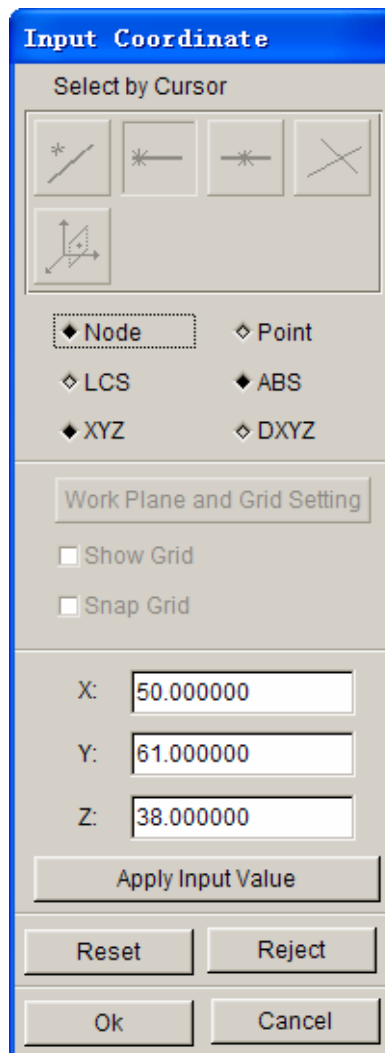


Figure 6.1.30 Selection of Tipping Center using INPUT COORDINATE dialog box

### 6.1.10.2 TIPPING WITH GEOMETRY (TOGGLE)

If the checkbox of this option is toggled on, eta/DYNAFORM will tip all the parts that are defined as DIE. All geometry in DIE including surfaces, lines and elements will be rotated and translated.



### 6.1.10.3 TIPPING WITH ALL PARTS (TOGGLE)

If the checkbox of this option is toggled on, eta/DYNAFORM will tip all the parts in the active database. All geometry including surfaces, lines and elements will be rotated and translated.

### 6.1.10.4 TIPPING CHECK

Upon completion of tipping operation, the user is recommended to apply the UNDERCUT and DRAW DEPTH functions to check the result.

- DRAW DEPTH (toggle)

This function displays a contour plot showing the draw depth of part relative to the binder. Different draw depths are displayed in graduated colors on the die. Draw depth is estimated with reference to the first point of punch/die contact. When the die/punch is closed, the first point on the blank to contact with the punch is locked. The rest of the part will be stretched from this point spreading outward.

- UNDERCUT (toggle)

This function displays a contour plot showing regions in the die with undercut. Red indicates severe under cut where the backdraft is less than 0°. Blue indicates a marginal area with a draw angle of approximately 1~3°. Green indicates a safe region with a draw angle that is greater than 3°. The user can set the threshold for marginal and severe backdraft.

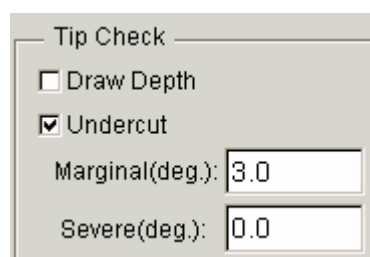


Figure 6.1.31 Tip Check Options

### 6.1.10.5 AUTO-TIPPING

This function automatically converts the die to stamping position by averaging all normal vectors of elements to minimize undercut and draw depth.



### 6.1.10.7 UNDO AND RESET TIPPING OPERATION

The UNDO function will cancel the last performed tipping operation, while the RESET function will cancel all tipping operations.

### 6.1.10.8 DEFINED TIPPING

This function is used to record and restore the tipping operation. The ADD function is used to record the tipping result. The REMOVE function is used to remove one record. The LOAD function is used to restore the tipping result to the selected record.

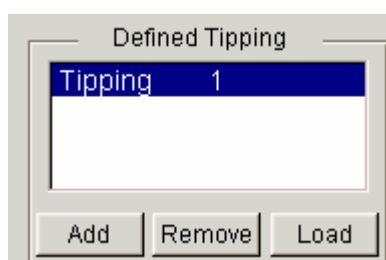


Figure 6.1.33 Defined Tipping field

### 6.1.11 OUTER SMOOTH

The functions provided in the OUTER SMOOTH dialog box are used to smooth the outer boundary of the selected part. As illustrated in Figure 6.1.34, there are four functions in the dialog box (ROLLER, MORPHING, CORNER SMOOTH, EXPAND).

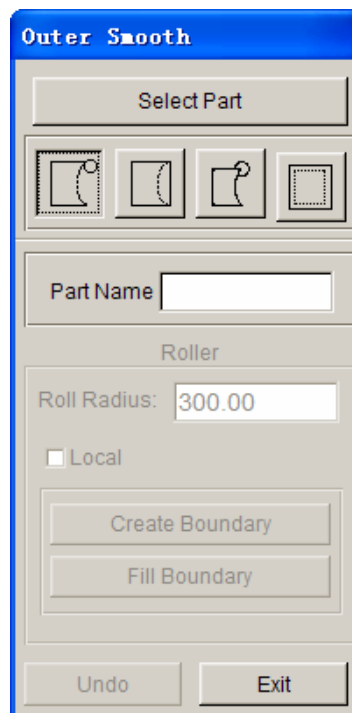


Figure 6.1.34 Outer Smooth dialog box

#### 6.1.11.1 ROLLER

This function allows the user to roll a cylinder with a defined radius along the boundary and fill the gaps between the part boundary and the cylinder. Once this function is selected, a dialog box as illustrated in Figure 6.1.35 is displayed. The default radius of the cylinder is 300 (mm) in the ROLL RADIUS input field.

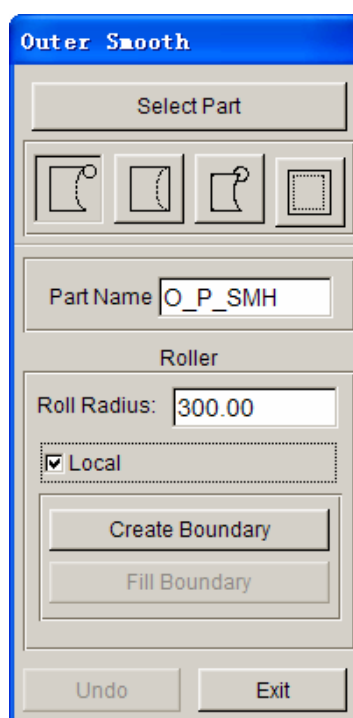


Figure 6.1.35 Roller dialog box

1. After the desired radius is entered, click on the CREATE BOUNDARY button to show the new boundary line with the cylinder rolled along the part boundary. If the result is acceptable, click on the FILL BOUNDARY button to fill the gap with elements and new surface. A new part will be created to include the elements and surfaces. The part will be automatically assigned as the DIE.
2. The UNDO function is used to remove the filled mesh in the gap and allow the user to go back to the original part. Figure 6.1.36 illustrates the result of a typical roller operation.

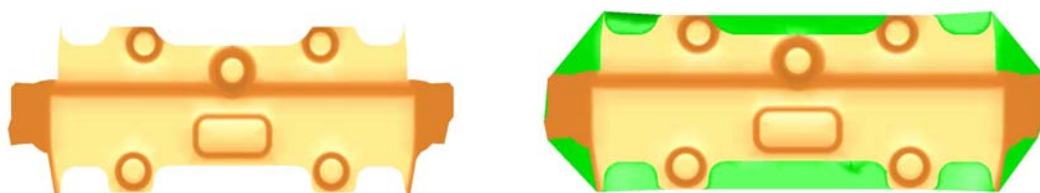


Figure 6.1.36 An illustration of a roller operation

3. Click on the CLOSE button to exit the function.

The LOCAL option allows the user to roll a cylinder with a defined radius along a selected boundary. eta/DYNAFORM will fill the gap between the part boundary and the cylinder surface to smooth the local boundary.

Click on the ROLLER button and select the LOCAL option. The default radius of the cylinder is set to 300 (mm) in the ROLL RADIUS input field. The user can input any desired PART NAME and ROLL RADIUS.

1. After the desired radius is entered, click on the CREATE BOUNDARY button to show the highlighted boundary line. eta/DYNAFORM requests the user to select two points on the boundary line (first and second) to identify the area to be rolled. Upon completing the point selection, the user needs to press the middle mouse button to continue the operation.
2. Once the middle mouse button is clicked, a smoother, virtual boundary will be highlighted over the original boundary. If the result is acceptable, click on the FILL BOUNDARY button to fill the gap. eta/DYNAFORM will create a new part with the given name to include the filled mesh and surfaces. In addition, this part will be automatically assigned as DIE. Fig. 6.1.37 illustrates the result of a typical roller operation using the LOCAL option.

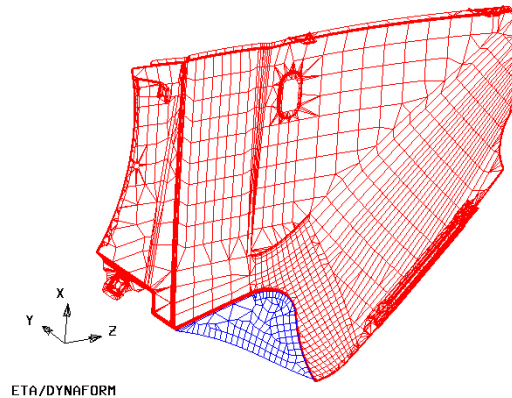


Fig. 6.1.37 An illustration of Outer Smooth/Roller using Local option

### 6.1.11.2 MORPHING

This function allows the user to locally modify the part mesh by morphing the boundary line of the part. Figure 6.1.38 illustrates dialog box of the MORPHING function.

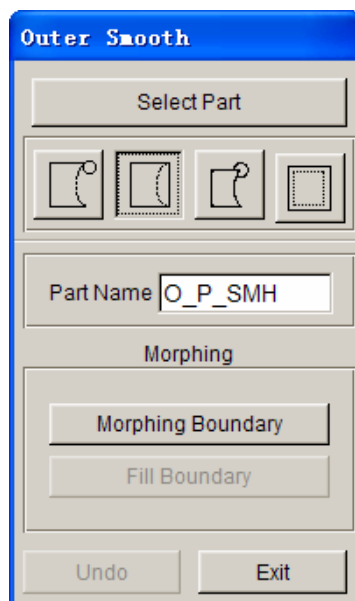


Figure 6.1.38 Morphing dialog box

1. Click on the MORPHING BOUNDARY button to highlight the boundary of die. Eta.DYNAFORM requests the user to select two points on the part boundary to define a morphing region. Figure 6.1.39 illustrates the selected region on the part boundary for morphing operation.

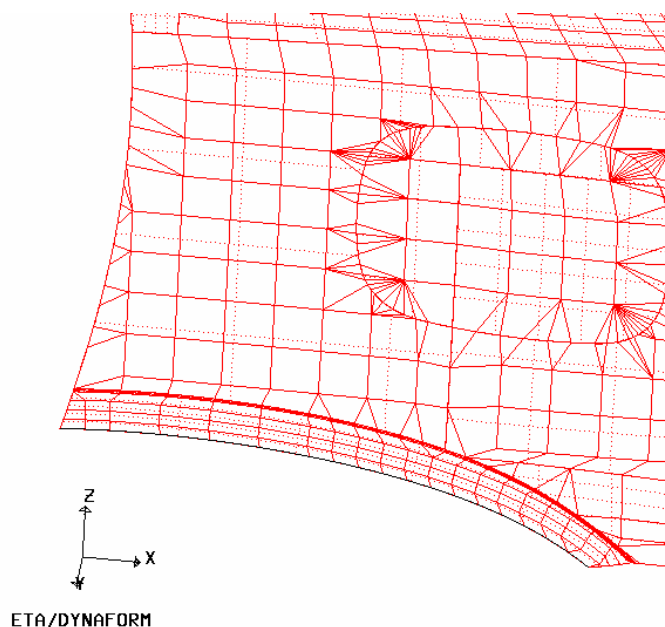


Figure 6.1.39 Select Morphing Boundary

2. After two points are selected, the CURVE EDIT dialog box as illustrated in Figure 6.1.40 is displayed. There are several options to allow the user to morph and smooth the boundary line.

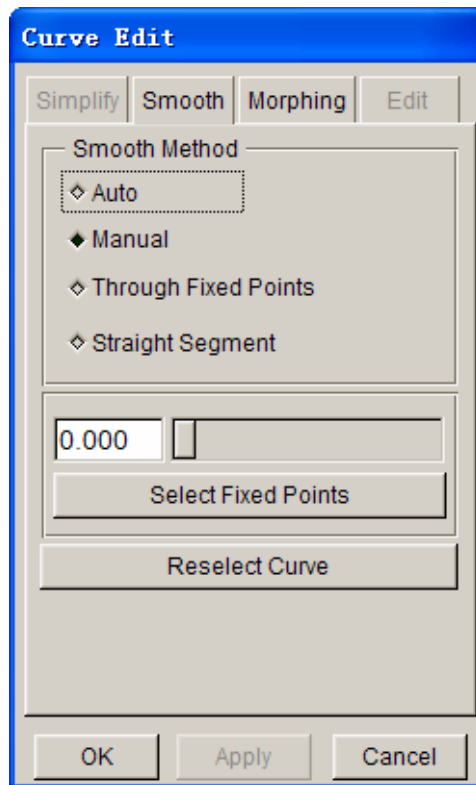


Figure 6.1.40 Curve Edit dialog box

About the detailed description of Curve Edit function, user can refer to the Section 6.2.1.

3. Click on the FILL BOUNDARY button to fill elements between the original boundary and the smoothed boundary. The result from a typical morphing operation is illustrated in Figure 6.1.41.

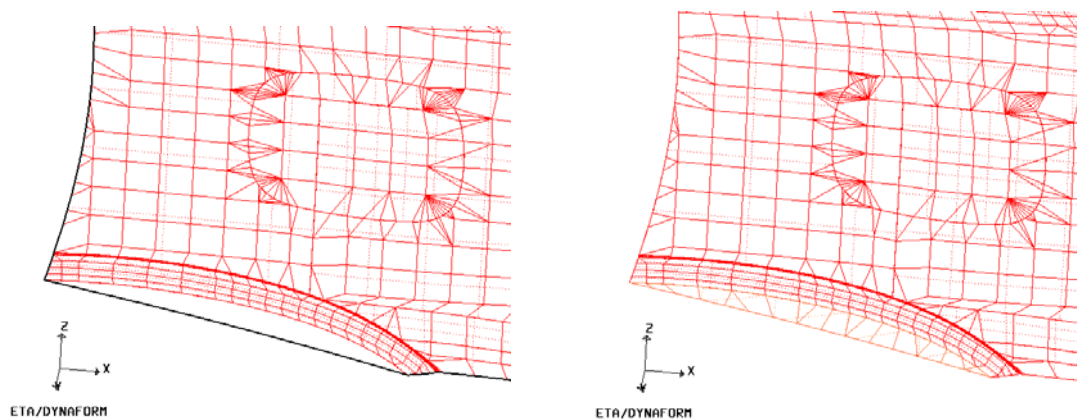


Figure 6.1.41 Morphing Result

4. The UNDO button is used to remove the elements from the gap, allowing the user to



return to the original boundary and keep the newly created part in IGS\_SMH. PART NAME enables the user to assign a name to the part; it will be automatically assigned as TOOL.

5. Click on the CLOSE button to exit the function .

### 6.1.11.3 CORNER SMOOTH

This function allows the user to smooth sharp corners on a part using a defined radius. eta/DYNAFORM will generate a set of elements in circular shape with the defined radius around the selected point.

When this function is selected, the dialog box as illustrated in Figure 6.1.42 is displayed. The default corner radius is 30 (mm) as displayed in the input field of CORNER RADIUS.

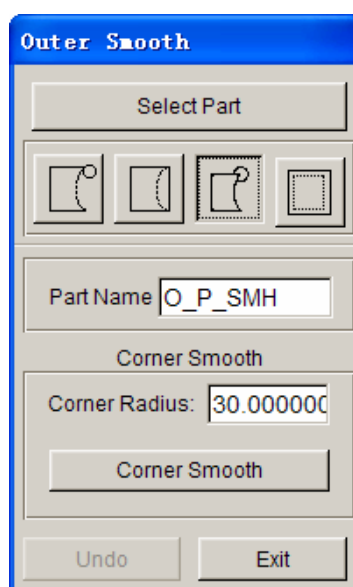


Figure 6.1.42 Corner Smooth dialog box

1. After entering the desired corner radius, click on the CORNER SMOOTH button to highlight the boundary line of the part such as illustrated in Figure 6.1.43.

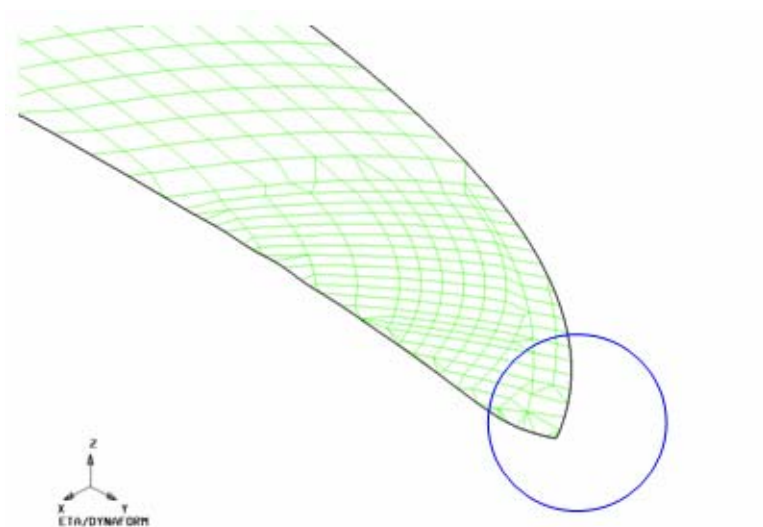


Figure 6.1.43 Sharp Corner

2. The user selects a node on the boundary for the corner smooth operation.
3. After a node is selected at a sharp corner, eta/DYNAFORM will generate a set of elements around the part boundary to create a circular shape to smooth the edge. Figure 6.1.44 illustrates a typical result of a corner smooth operation.

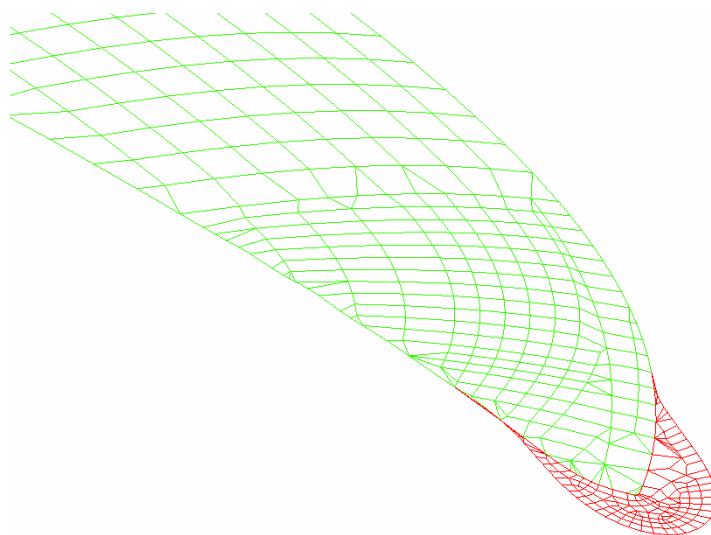


Figure 6.1.44 Corner Smooth

4. The UNDO button is used to remove the elements from the smoothed corner and allowing the user to obtain the original boundary.

5. Click on the CLOSE button to exit the function.

#### 6.1.11.4 EXPAND

The EXPAND function is used to extend the part boundary with elements. When this function is selected, dialog box as illustrated in Figure 6.1.45 is displayed. The value in the input field of EXTENSION is the default extension value for the active part. The user can type in any positive value.

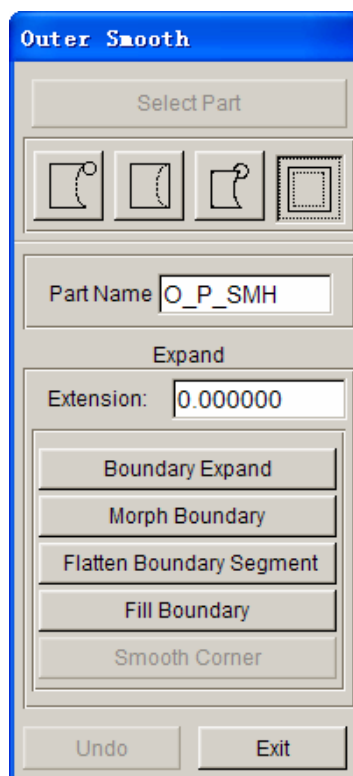


Figure 6.1.45 Boundary Expand dialog box

1. Click on the BOUNDARY EXPAND button to highlight the boundary of extension as illustrated in Figure 6.1.46a. The default extension value is calculated from the average element size. The user can input other value following by clicking the BOUNDARY EXPAND button once again to modify the default value.
2. Select the MORPH BOUNDARY button to morph the boundary line. Refer to Section 6.5.1 LINE MORPHING for description of this function.
3. The FLATTEN BOUNDARY SEGMENT function allows the user to flatten line segment between two selected points on the boundary line.

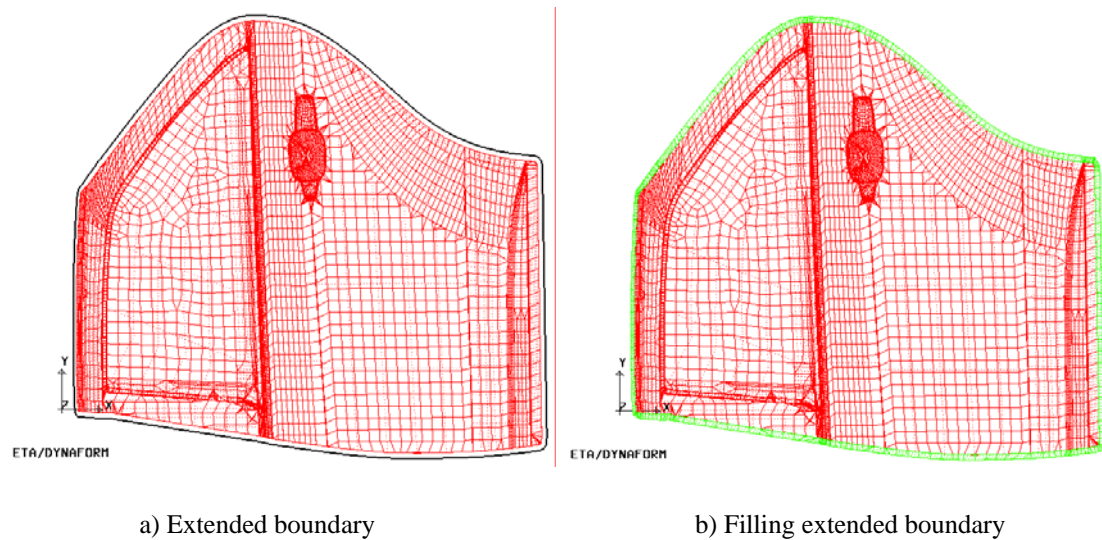


Figure 6.1.46 An illustration of the Boundary Expand operation

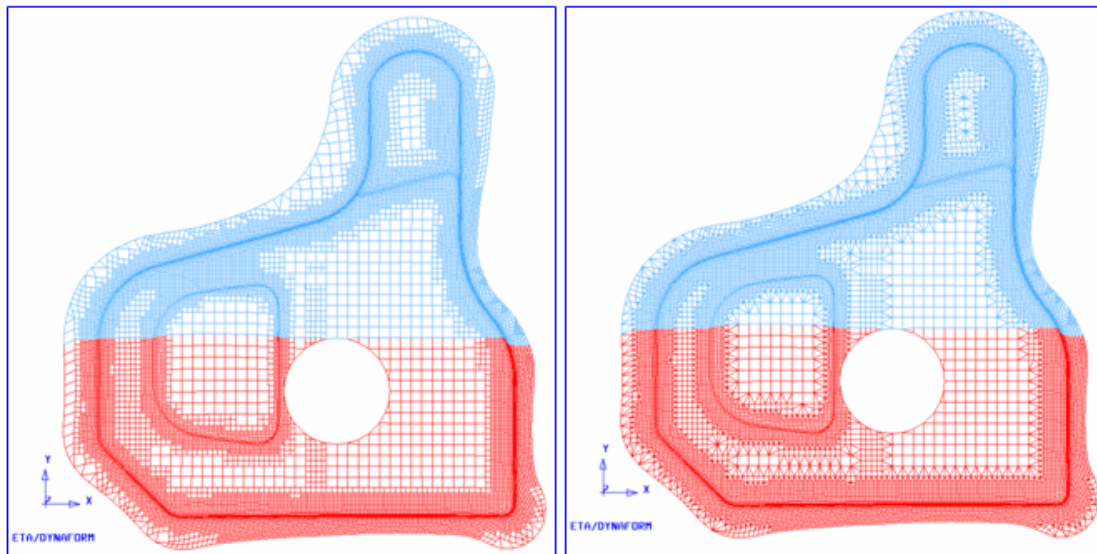
4. When the morphed boundary is accepted, click on the FILL BOUNDARY button to generate the mesh between the original boundary and the extension boundary. An illustration of extended boundary with filled elements is illustrated in Figure 6.1.46b.
5. The SMOOTH CORNER function allows the user to smooth a selected sharp corner.

**NOTE:** A sharp angle is determined by two adjacent boundary elements when the angle between two boundary edges is greater than  $30^\circ$ .

6. Click on the UNDO button to remove the generated mesh and exit the function.

### 6.1.12 TOPOLOGY REPAIR

This function allows the user to auto repair topological structure which is not integrated. After selecting the option, the program will pop up the dialog box in which the user can select the part to repair. The user can compare the original topological structure with the repaired as shown Figure 6.1.47.



a) Original topology

b) Repaired topology

Figure 6.1.47 Auto topology repair

## 6.2 BINDER

The functions provided in the BINDER menu are used to generate various types of binder shapes. Figure 6.2.1 shows the types of binder available in the BINDER menu.

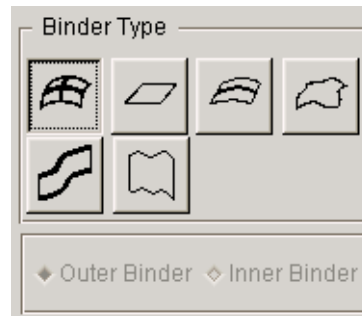


Figure 6.2.1 Binder Types

1. **TWO-LINE BINDER**  
Use two orthogonal cross lines to define the binder shape.
2. **FLAT BINDER**  
Use a flat rectangular surface as binder.
3. **CONICAL BINDER**  
Use a conical surface with two radii to define the binder shape.
4. **BOUNDARY LINE BINDER**  
Use the part boundary line or user-selected/created line to define the binder.
5. **FLANGE BINDER**  
Use the flanges on the part to define the binder shape.
6. **FREE FORM BINDER**  
Use the binder section lines from selected locations on the screen to define the binder.

The outer or inner binder is associated with the outer or inner addendum; Only one outer binder and addendum pair is allowed in a typical die face design using DFE, while several inner binder and inner addendum pairs can be created according to product design.

A detailed description of each function in BINDER dialog box is provided in the following sections.

### 6.2.1 TWO-LINE BINDER

The TWO-LINE BINDER function is used to create a binder with curvatures in both the X and Y directions. The dialog box is illustrated in Figure 6.2.2.

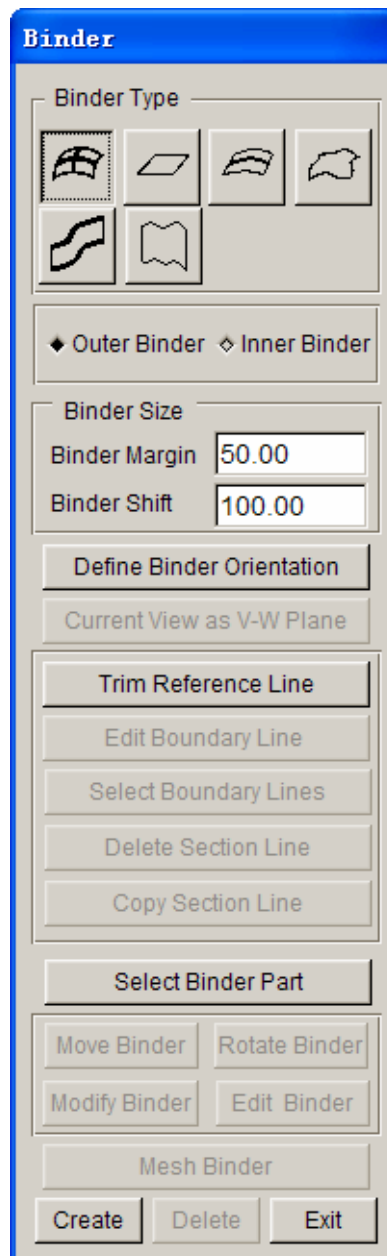


Figure 6.2.2 Two-line Binder dialog box

The BINDER MARGIN and BINDER SHIFT parameters are used to control the binder size and position relative to the part, respectively.

- **BINDER MARGIN**

This function allows the user to expand the binder size. The value of the BINDER MARGIN is the distance between the boundary of the original binder and the expanded binder. Please see the Figure 6.2.3.

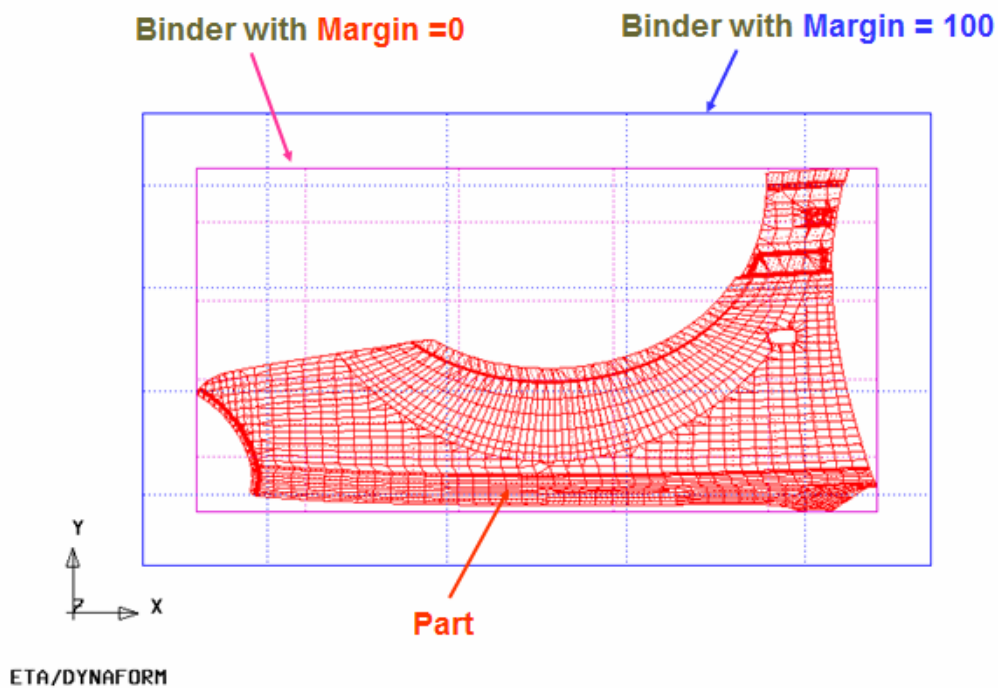


Figure 6.2.3 Binder Margin

- **BINDER SHIFT**

This function is used to adjust the shortest vertical clearance between the part edge and the binder surface. If positive BINDER SHIFT is entered, the binder will be shifted along the -Z axis and vice versa. The BINDER SHIFT directly affects the draw depth of a part. Please see the Figure 6.2.4.

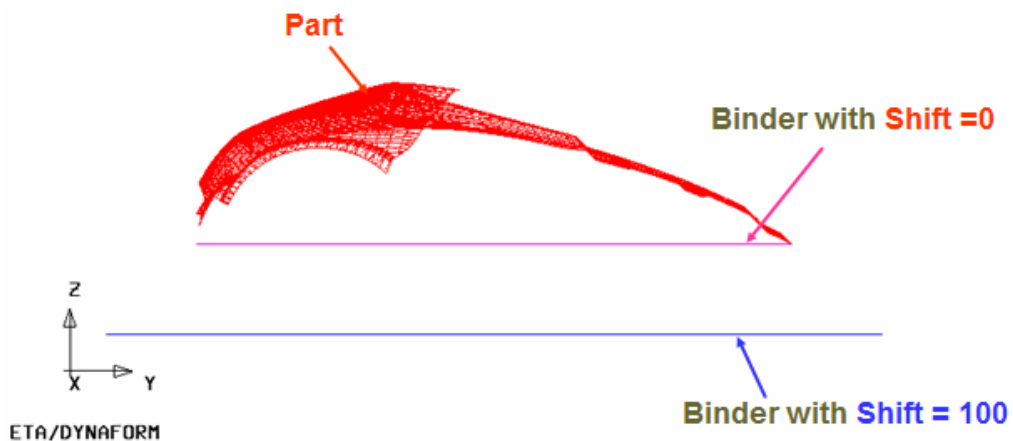


Figure 6.2.4 Binder Shift



- **DEFINE BINDER ORIENTATION**

The DEFINE BINDER ORIENTATION function controls the U-V direction of the binder surface.

When the button is clicked, the part is displayed in the X-Y view. The POINT OF FIRST CONTACT is highlighted with a white circle and the following message is printed in the message prompt window:

***CLICK AND HOLD THE LEFT MOUSE BUTTON AT THE CROSS HAIR CENTER. MOVE THE MOUSE TO ROTATE. RELEASE THE MOUSE BUTTON WHEN DONE***

As illustrated in Figure 6.2.5, two orthogonal cross lines is created. The line with an arrowhead indicates U-axis of the binder surface. Click and hold left mouse button, following by moving the mouse to obtain a desirable orientation of the U-V axes.

- **TRIM REFERENCE LINE**

The shape of the binder is determined by the following factors: reference point, orientation of the U-V axes, length of the U-V axes, and the part geometry. If some of the details on the part are not acceptable during creation of binder shape, the user can truncate the reference line to adjust the curvature of the binder. Figures 6.2.5-6.2.7 illustrates binders created using different lengths of U-V axes.

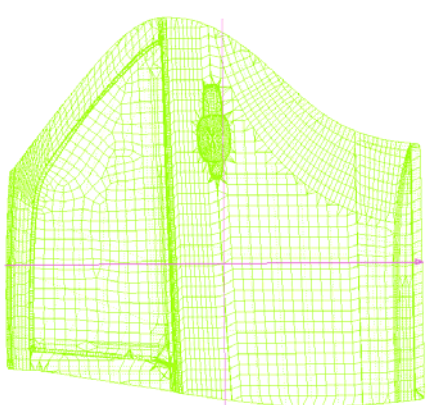


Figure 6.2.5 Two Control Lines

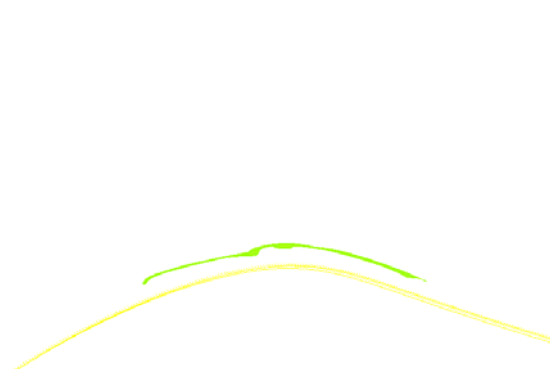


Figure 6.2.6 Two Line binder (before trimming)

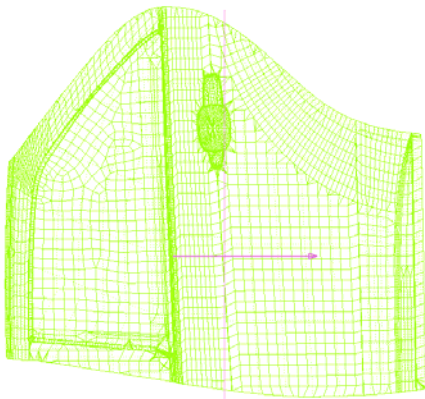


Figure 6.2.7 Trim Reference Line

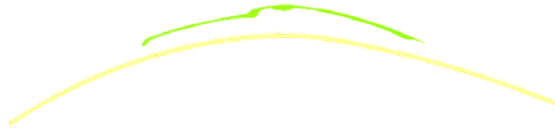


Figure 6.2.8 Two Line Binder From Trimmed Reference Line

The following steps are needed to create a binder:

1. Enter the BINDER MARGIN and BINDER SHIFT.
2. Define the proper orientation of the U-V direction.
3. Adjust the binder shape by using the TRIM REFERENCE LINE function.
4. Click the CREATE button at the bottom of the dialog box. eta/DYNAFORM generates the binder surface and stores it in a new part named C\_BINDER. The part is automatically assigned as TOOL (binder). Refer to Section 10.2 for a detailed explanation on the TOOL part.
5. If a binder has already been created in the C\_BINDER part, a DYNAFORM QUESTION dialog box as illustrated in Figure 6.2.9 is displayed.

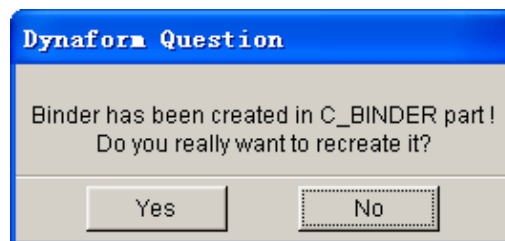


Figure 6.2.9 DYNAFORM QUESTION dialog box

**YES** – Override previously created binder with the current definition.

**NO** – Keeps the previously created binder.

- **SELECT BINDER PART**

When more than one binder is available in the database, the user can click the SELECT BINDER PART button to select a specific binder from the SELECT BINDER dialog box as illustrated in Figure 6.2.10.

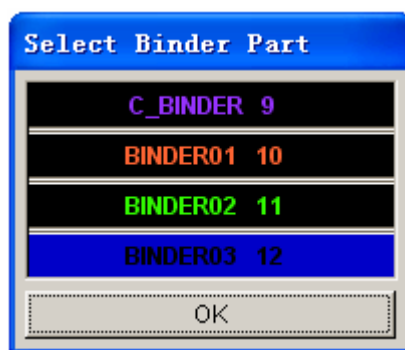


Figure 6.2.10 Select Binder dialog box

Once the binder is created, the user can edit binder using the editing functions as illustrated in Figure 6.2.11.

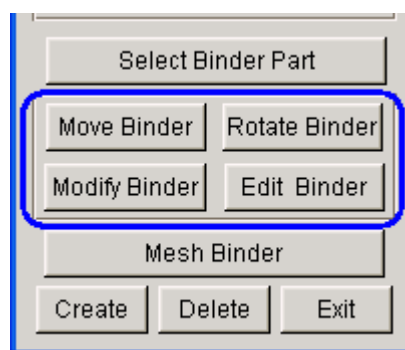


Figure 6.2.11 Binder Editing Options

The editing functions are described as following:

- **MOVE BINDER**

This function allows the user to adjust relative distance between the binder and the part. The relative distance will affect the draw depth of the part. Click on the MOVE BINDER button to display a dialog box as illustrated in Figure 6.2.12.

The user can select the following methods to move the binder:

1. Select MOVE DIRECTION (U, V or W) and enter the translation distance in the input field of U, V and W. Click on the APPLY button to move the binder along the selected direction. The REVERSE OPERATION option is used to reverse the translation of binder.
2. Translate the binder by dragging the SLIDER bar as illustrated in Figure 6.2.10. If the bar is slid to the right, the binder will be translated in the positive direction along the selected axis and vice versa. Click on the APPLY button to accept the new binder position.

The total binder travel will be displayed in the input field of BINDER TRAVEL. The total binder travel is dynamically updated as the user moves the slider.

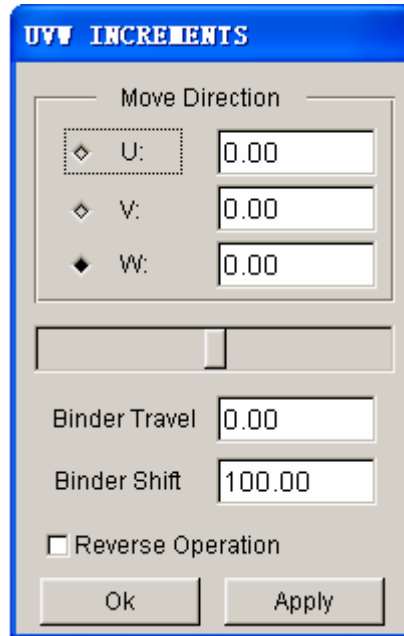


Figure 6.2.12 Move Binder dialog box

- **ROTATE BINDER**

This function allows the user to rotate the binder to an orientation relative to the part.

1. Click on the ROTATE BINDER button to display a dialog box as illustrated in Figure 6.2.13. By default, the center of binder is set as the center of rotation with U, V, W axes parallel to X, Y, Z axes of global coordinate system, respectively. The user can define the center of rotation by click on the CENTER OF ROTATION button, following by picking a location using mouse cursor.

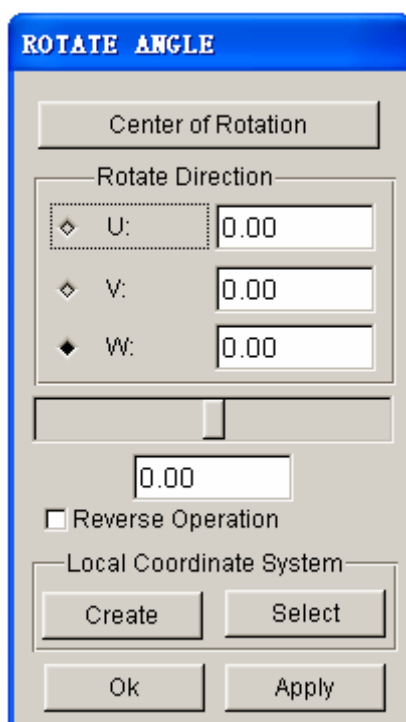


Figure 6.2.13 Rotate Binder dialog box

2. After the center of rotation is defined, the user can select axis of rotation by toggle on one of the three toggle boxes for U, V and W. The angle of rotation is entered in the input field, following by clicking on the APPLY button to rotate the binder.
3. The user is given an option to rotate the binder by moving the SLIDER bar to either left or right. Moving the slider to the left will rotate the binder in clockwise direction, and vice versa. While moving the slider, the user can observe the dynamic view of binder from the display area. The amount of rotation angle is recorded on the box right below the SLIDER bar as illustrated in Figure 6.2.11.
4. The user can also create a LCS which U, V and W axes are parallel to the X, Y, Z of global coordinate system. Click on the CREATE button to display a dialog box as illustrated in Figure 6.2.14 to define the LCS. Select a point as the origin and click on the OK button to create the LCS. Click on the SELECT button to display the SELECT LCS dialog box, following by clicking on the C.S.ID button to select the appropriate identification number of the LCS.
5. Once the appropriate LSC is selected, the user can select the axis of rotation by toggle on U, V, or W. The angle of rotation is entered in the input field, following by clicking on the APPLY button to rotate the binder. The user can also rotate the

binder by moving the slider bar to either left or right.

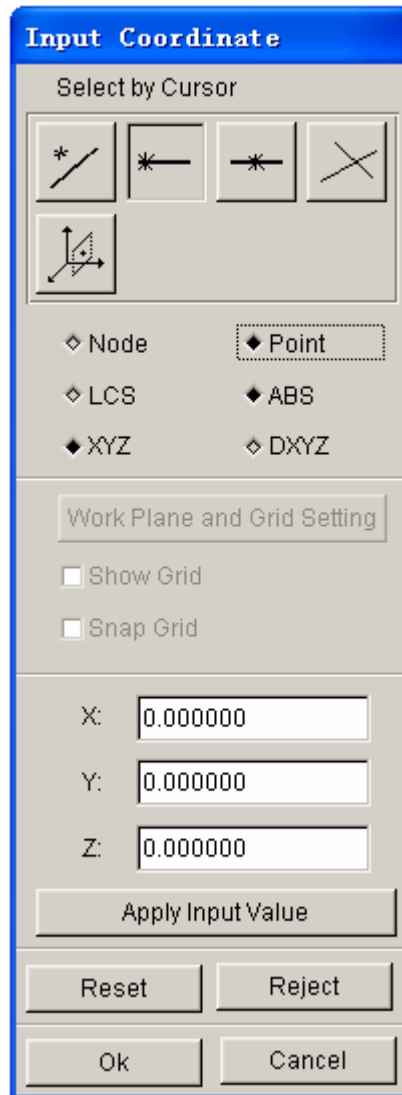


Figure 6.2.14 INPUT COORDINATE dialog box for selecting origin of the LCS

6. If the REVERSE OPERATION option is selected, and the binder will be rotated about the defined axis of rotation in the opposite direction after clicking the APPLY button.
7. Click on the OK button to dismiss the dialog box.

- **MODIFY BINDER**

This function provides the user with four options to change the binder shape by modifying one of two cross section lines which defines the binder shape.

1. Click on the MODIFY BINDER button, following by selecting one of the section

lines from the display area. Two cross section lines which define the binder shape are illustrated in Figure 6.2.15.

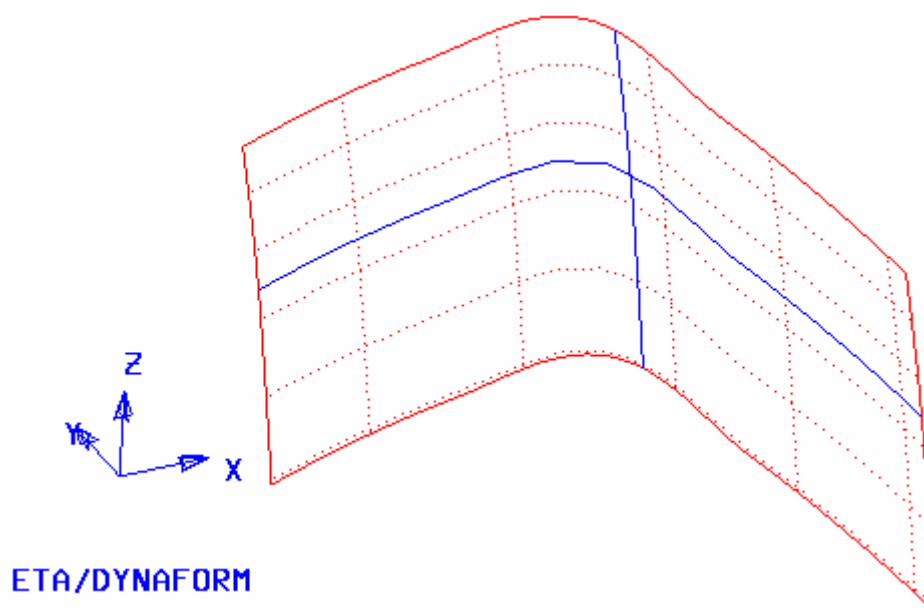


Figure 6.2.15 An illustration showing two section lines for Edit Binder operation

2. After a section line is selected, the CURVE EDIT dialog box is displayed. There are four option tabs available to the user for editing the section line.

1. **SIMPLIFY**

This function reduces the number of points of the selected line. The dialog box is illustrated in Figure 6.2.16.

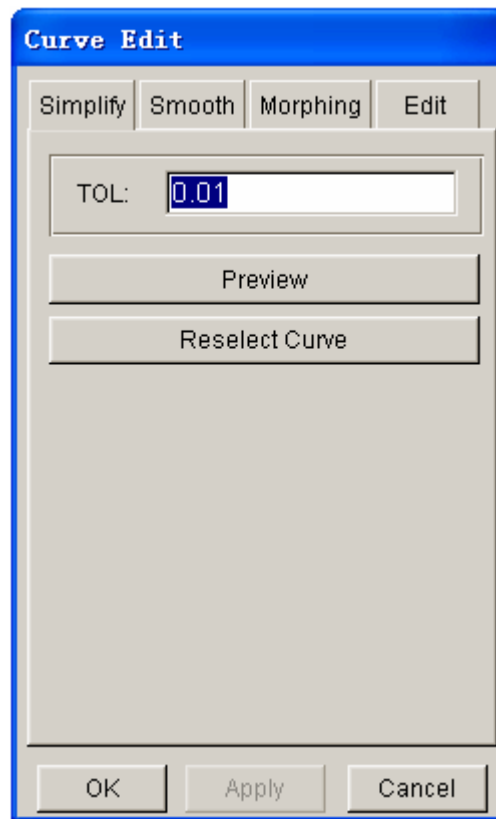


Figure 6.2.16 Simplify Curve dialog box

#### **TOL**

The criteria used to simplify the curve. If the distance between two adjacent points is less than the defined tolerance, one of the points will be removed from the curve.

#### **PREVIEW**

This function displays the new section line after some points are removed.

#### **RESELECT CURVE**

This function enables the user to select the other section line.

#### **APPLY**

The function completes the operation.

#### **OK**

Accept the operation and dismiss the dialog box.

#### **CANCEL**

Abort the operation and dismiss the dialog box.



## 2. SMOOTH

This function enables the user to smooth the selected section line. The dialog box is illustrated in Figure 6.2.17. Three methods can be utilized to smooth the section line:

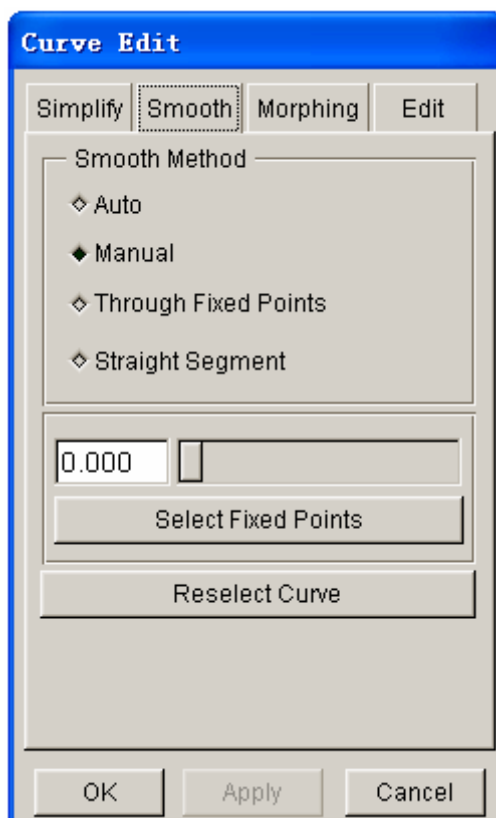


Figure 6.2.17 Smooth Curve dialog box

### AUTO SMOOTH

If this function is toggled ON, the program will automatically smooth the selected section line.

### MANUAL SMOOTH

If this function is toggled on, the slider and SELECT POINT button will be activated. The user can enter any value between 0 and 1.0 in the input field next to the slider as the smooth coefficient and click on the APPLY button to smooth the curve. The user can also drag the slider to smooth the curve in dynamic mode. eta/DYNAFORM will dynamically display the smoothed curve as the slider is dragged, while simultaneously updates the smooth coefficient next to the slider. Click on SELECT POINT to select constraint point(s) on the curve.

Other points on the curve will be smoothed.

### THROUGH FIXED POINTS

When this function is toggled ON, the SELECT POINT button will be activated. Click on the SELECT POINT button to highlight all the points on the selected curve. A new spline curve passing through the selected point(s) will be created.

### STRAIGHT SEGMENT

This function enables user to smooth the curve through replacing the segment that user selected with straight line. If user toggles on this option, the Select Segment button is activated. User can click this button to select two points in curve. Click the Apply button after selected two points to change the curve to straight line.

## 3. MORPHING

This function allows the user to modify the section line using four different morphing methods. Figure 6.2.18 illustrates the characteristics and results of each morphing method.

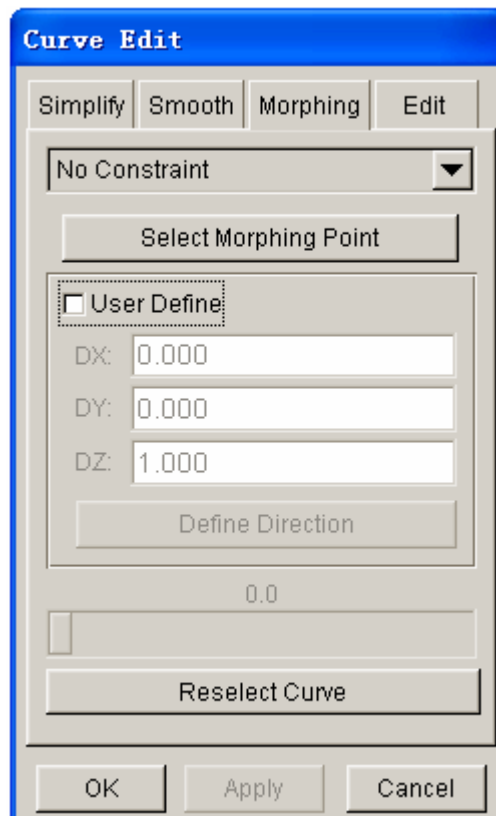
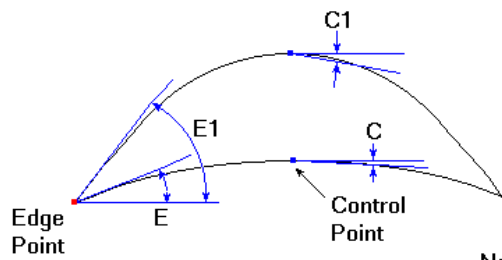


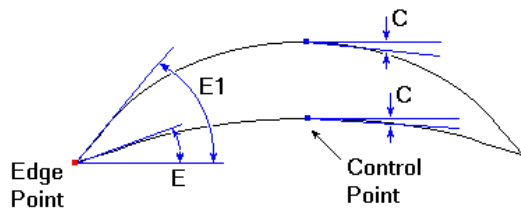
Figure 6.2.18 Morphing Curve dialog box

DIFFERENT TYPES OF SURFACE SECTION FOR MORPHING

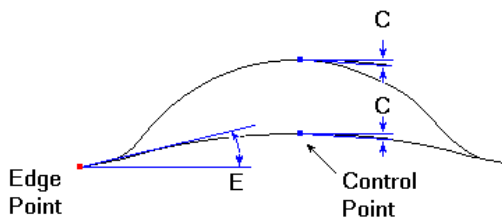


Type 1 - No constraint. Curvature at Control Point and Edge Point change. C1 not equal to C. E1 not equal to E. This is the shape currently used in all morphing types. No very suitable.

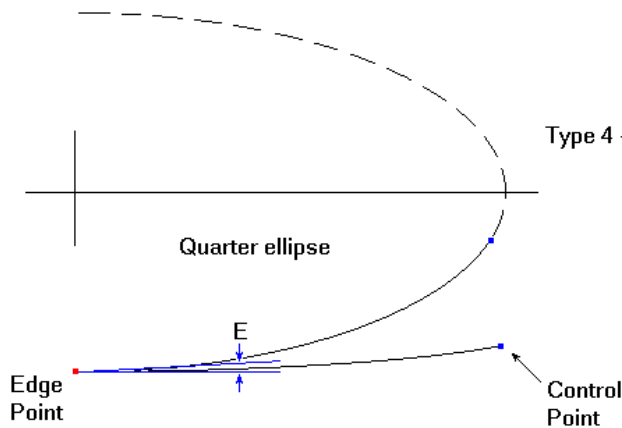
Note: C - Angle between the element plane and horizontal plane at Control Point. C1 - Angle C after morphing. E - Angle between the element plane and horizontal plane at Edge Point. E1 - Angle E after morphing.



Type 2 - Constraint at Control Point. Curvature at Control Point does not change. Curvature at Edge Point changes. E1 not equal to E. This type can be an option for Edge morphing and Interior morphing. Not for Corner Morphing.



Type 3 - Constraint at Control Point and Edge Point. Curvature at Control Point and Edge Point do not change. This should be the default shape for Edge Morphing and Interior morphing. It can be an option for Corner Morphing.



Type 4 - Constraint at Edge Point. Curvature at Edge point does not change. The shape is part of a quarter ellipse. This should be the default shape for Corner Morphing.

Figure 6.2.19 Morphing Methods

**NO CONSTRAINT**

There is no constraint imposed on the section line when it is morphed. The morphed shape will assume a natural spline curve. Refer to type 1 as illustrated in Figure 6.2.19 for the shape of section line after morphing.

**KEEP C ANGLE**

The tangent at the control point on the section line is constrained. The morphed shape will maintain the original tangent direction at the control point of the section line. Refer to type 2 as illustrated in Figure 6.2.19 for the shape of section line after morphing.

**KEEP E ANGLE**

The tangent at both end points on the section line is constrained. The morphed shape will maintain the original tangent direction at the end points of the section line. Refer to type 3 as illustrated in Figure 6.2.19 for the shape of section line after morphing.

**KEEP CONVEX**

Maintain the convex edge by changing the curvature of the section line.

Figure 6.2.18 illustrates the dialog box of the MORPHING function. Click on the SELECT MORPHING POINT to select the control point. eta/DYNAFORM will highlight the points on the section line and request the user to select one control point. A vector is plotted at the morphing point to indicate the morphing direction. The user may move the cursor to see the new shape of the section line in real time. Click on the RIGHT MOUSE BUTTON to accept the morphed shape. Click on the APPLY button to update the binder shape according to the morphed section line. eta/DYNAFORM calculates a default morphing direction that is normal to the section line and the binder surface. The user may click on the USER DEFINE option to enter the desirable vector for the morphing direction.

**4. EDIT**

This function provides a method to edit the selected section line by modifying the points on the line. As illustrated in Figure 6.2.20, the options for editing the section line are:

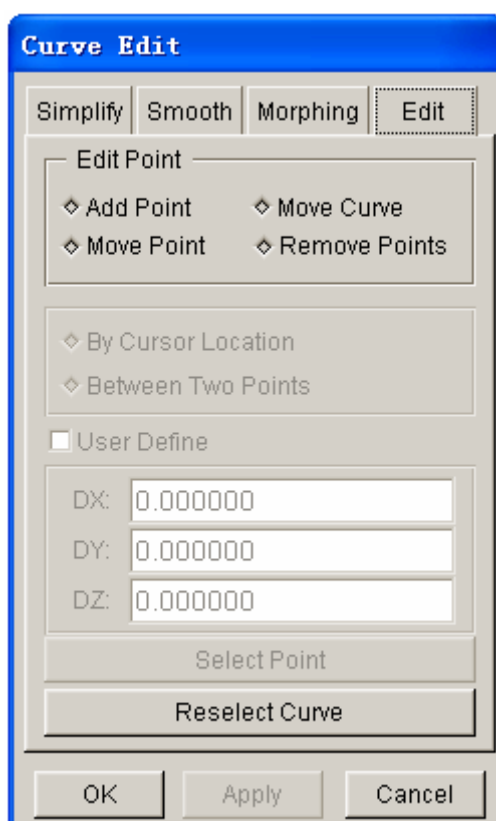


Figure 6.2.20 Edit Curve dialog box

### ADD POINT

When this option is selected, the BY CURSOR LOCATION and BETWEEN TWO POINTS buttons are activated. The user may toggle on the BY CURSOR LOCATION button to add point on any location on the section line. Toggle on the BETWEEN TWO POINTS option, and click on the SELECT POINT button. eta/DYNAFORM displays existing points on the section line and request the user to select two adjacent points. After two adjacent points are selected, the POINT NUMBER input field is activated to allow the user to enter the number of points to be added between two selected points. If the second selected point is not adjacent to the first one, the following an error message is printed in the message prompt window:

***INVALID POINT SELECT, REJECTED***

### MOVE CURVE

This function enables the user to move a selected section line. The binder surface will be moved along with the section line. Toggle the option on and the

program will activate three data windows next to DX, XY and DZ. The user may enter any three displacement vectors with respect to the global X, Y and Z direction. Click APPLY button to move the section line and the binder.

### MOVE POINT

This function allows the user to move a point on the section line in order to move the binder accordingly. If the option is selected, the SELECT POINT button and three input field next to DX, DY and DZ will be activated. The user may click on the SELECT POINT button to select a point on the section line, following by entering the displacement value with respect to the global X, Y and Z direction. Click on the APPLY button to move the points. The binder shape will be modified according to the new shape of the section line.

### REMOVE POINT

This function enables the user to remove the control points from the selected section line. If the option is selected, the SELECT POINT button is activated to enable the user to select the point(s) on the section line. Click on the middle mouse button to complete the point selection. Refer to Figure 6.2.21 for an illustration of the control points.

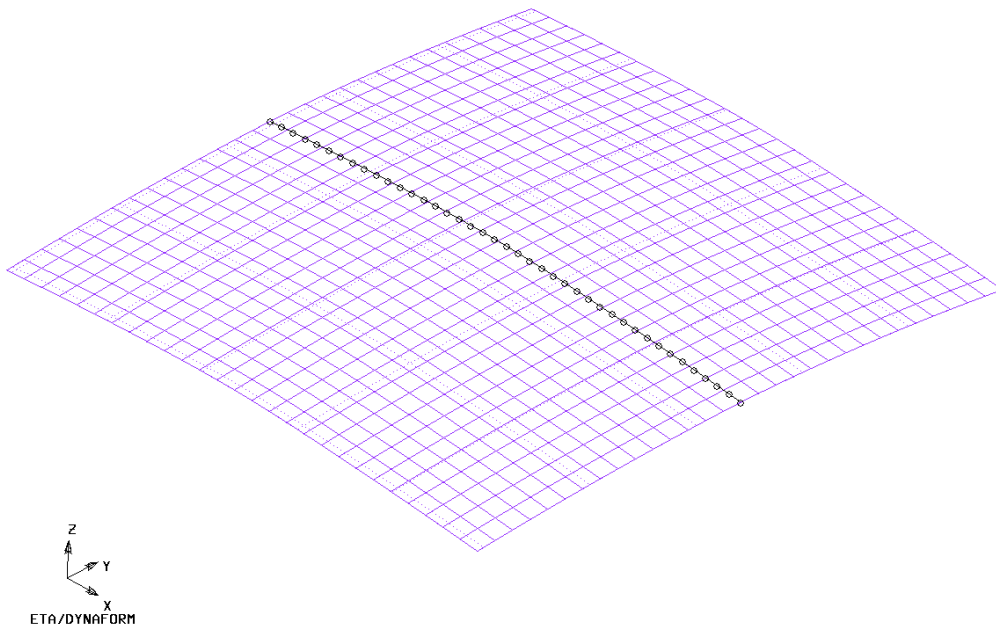


Figure 6.2.21 Control Points On Section Curve

- **MESH BINDER**

This function is used to generate the binder mesh using the binder surface. A dialog box as illustrated in Figure 6.2.22 is displayed.

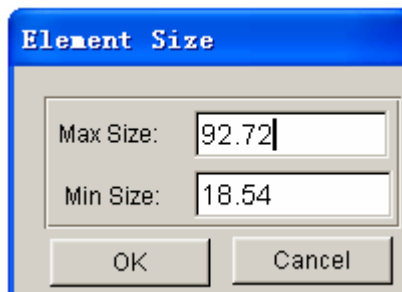


Figure 6.2.22 Element Size dialog box

eta/DYNAFORM calculates the default maximum and minimum element size of the mesh based on the shape and size of the binder surface. The user may enter the desirable value in the input fields. MAX SIZE controls the maximum element size of the mesh, while MIN SIZE controls the minimum element size of the mesh.

- **CREATE**

This function creates a binder surface and stores it in the part named C\_BINDER.

*NOTE: If there is an existing C\_BINDER part in the current database, eta/DYNAFORM creates a part named BINDER01. Subsequently, if additional binders are created, BINDER $n+1$  will be created. eta/DYNAFORM recognizes the binder created from the binder menu.*

- **DELETE**

Remove all binder surfaces and mesh from the database.

- **EXIT**

Close the BINDER dialog box.

## 6.2.2 FLAT BINDER

This function is used to create a flat binder. As illustrated in Figure 6.2.23, two options are used to define the orientation of a flat binder.

## DEFINE BINDER ORIENTATION

Refer to Section 6.2.1 for a detailed description defining binder orientation.

## SELECT 3 NODE/POINT

This function enables the user to select three nodes to define binder orientation. The first two nodes are used to define the U direction of the binder surface, while the third node completes definition of the flat binder surface.

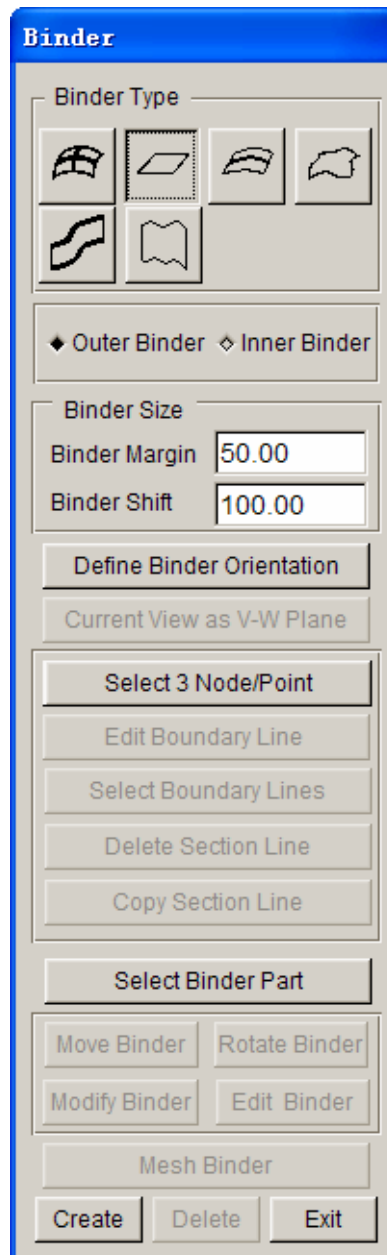


Figure 6.2.23 Flat Binder dialog box



The UV line direction of binder surface can be defined by using the DEFINE BINDER ORIENTATION function. All other functions are similar to those described in Section 6.2.1.

### 6.2.3 CONICAL BINDER

This function is used to create a conical binder surface using two arcs. The radius defined at each end of the binder determines the conical shape. Once the function is selected, the input field of RADIUS 1 and RADIUS 2, as illustrated in Figure 6.2.24, are activated.

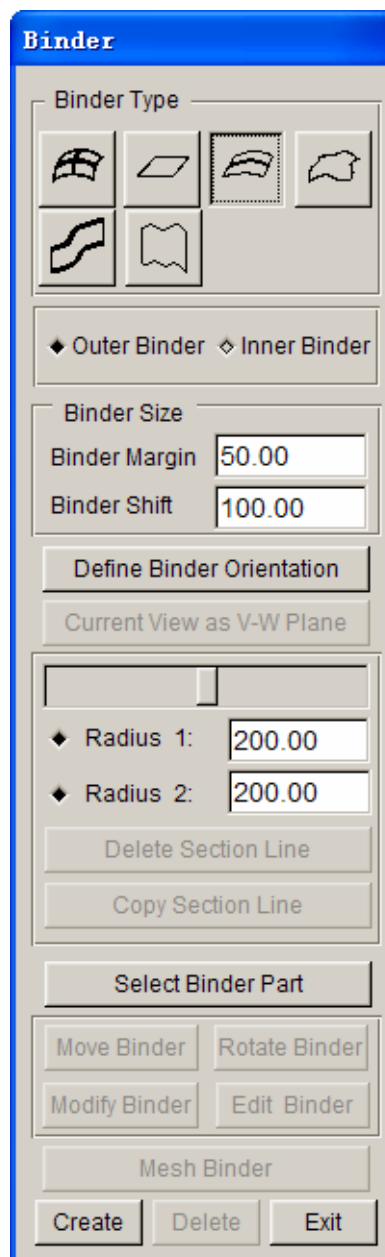


Figure 6.2.24 Conical Binder dialog box

The user can drag the slider to modify the RADIUS1 and RADIUS2, simultaneously or separately. Click on the radio button located next to Radius 1 and Radius 2 to activate the slider effect. Radius 1 is the starting radius at the tail end of the U direction that is marked with an arrowhead. Radius 2 is the end radius on the opposite end of the binder. eta/DYNAFORM dynamically displays the binder shape as the user drag the slider to modify the radius. Figure 6.2.25 shows a typical conical binder.

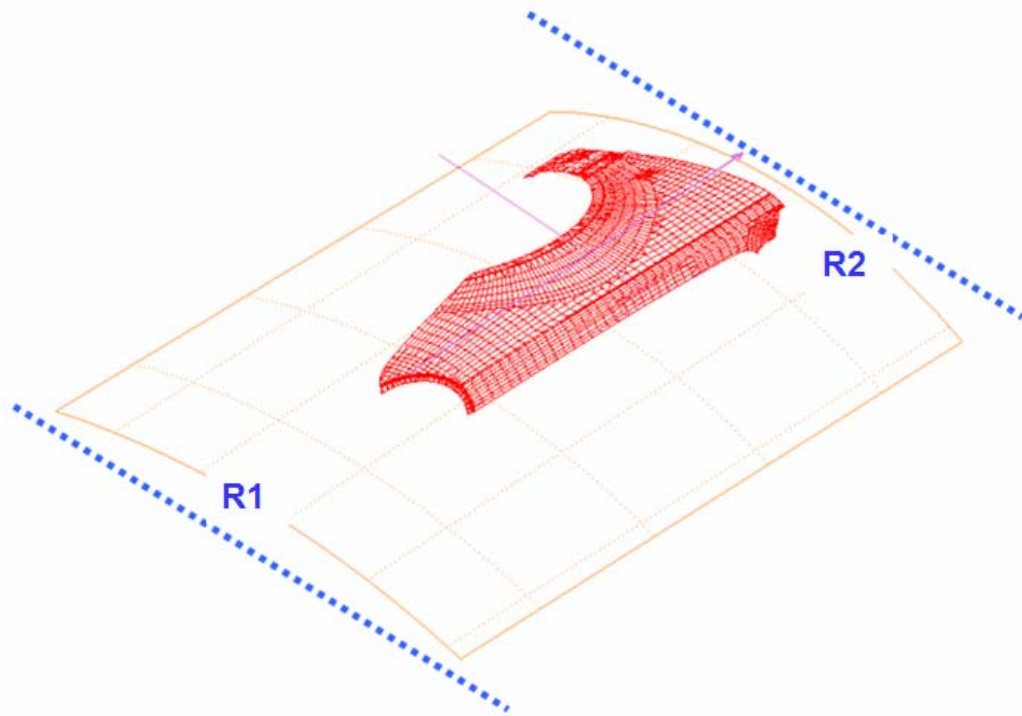


Figure 6.2.25 An illustration of generation of conical binder surface using two arcs

**NOTE:** eta/DYNAFORM calculates the default conical Radius 1 and conical Radius 2 based on the size and shape of the part.

#### 6.2.4 BOUNDARY LINE BINDER

This function generates a binder surface using the boundary of part. The dialog box is illustrated in Figure 6.2.26.

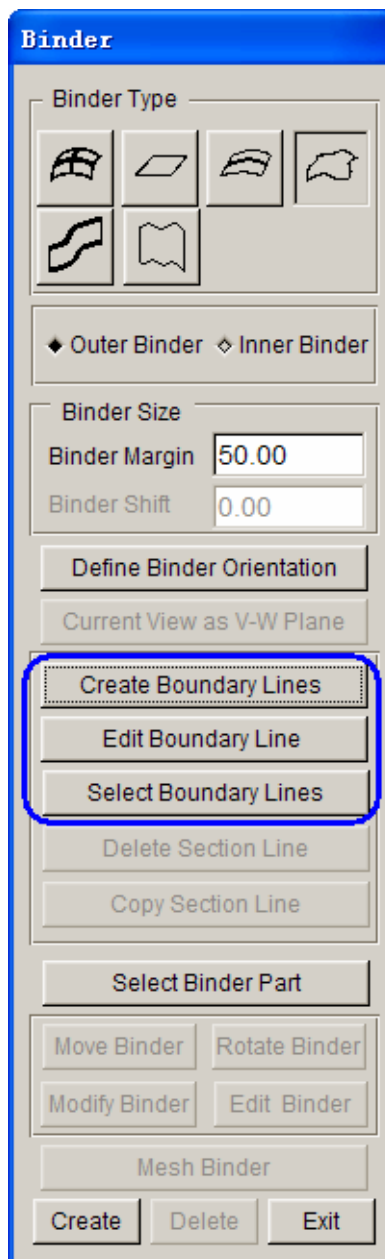


Figure 6.2.26 Boundary Line Binder dialog box

The following functions are used to generate the Boundary Line binder.

#### DEFINE BINDER ORIENTATION

This function defines the UV-direction of the binder surface. The line with the arrow controls the surface shape and must be parallel to the direction of the part boundary. Figure 6.2.27 shows a typical orientation of the U-V axes relative to the part.

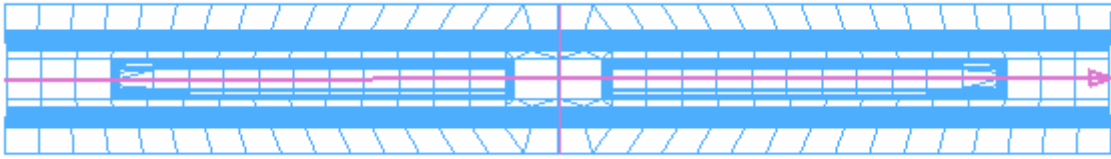


Figure 6.2.27 Define Binder Orientation

### CREATE BOUNDARY LINES

This function generates boundary lines of the part. The boundary lines are kept in the current part. Figure 6.2.29 illustrates typical boundary line of a part.

### EDIT BOUNDARY LINE

The functions in this menu are used to edit the boundary lines. The dialog box is illustrated in Figure 6.2.28. The description of edit line functions is discussed in Section 5.1.

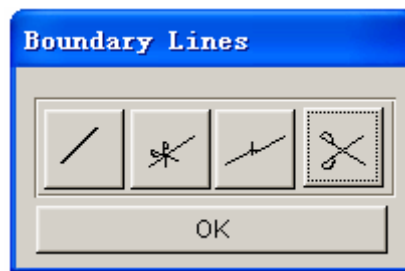


Figure 6.2.28 Edit Boundary Line dialog box

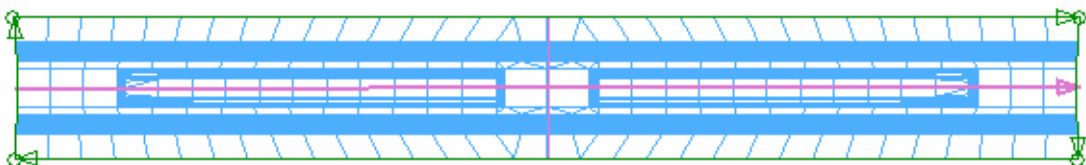


Figure 6.2.29 Created Boundary Line

### SELECT BOUNDARY LINE

This function is used to select the desired boundary lines to generate binder surface. The selected line will be highlighted such as illustrated in Figure 6.2.30. Click on the CREATE button to generate the binder surface. The generated binder is illustrated in Figure 6.2.31.

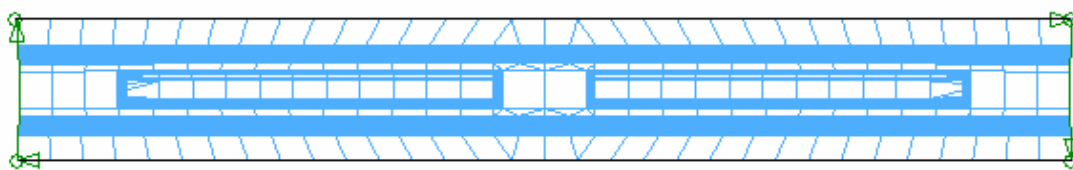


Figure 6.2.30 Selected Boundary Line

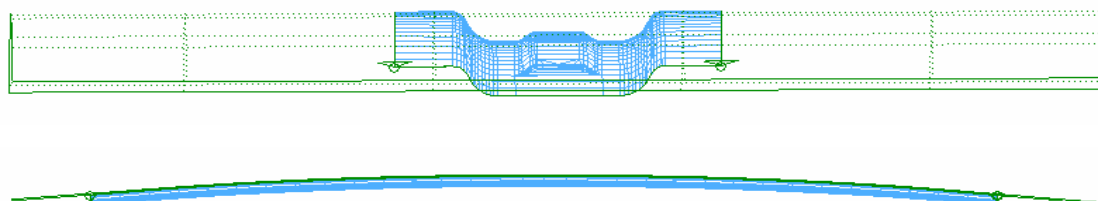


Figure 6.2.31 Created Binder

If the binder is not acceptable, the user may create another binder by editing the boundary.

For a detailed description of other editing functions, refer to Section 6.2.1.

### 6.2.5 FLANGE BINDER

This function is used to generate a binder surface based on the character lines of flanges. The dialog box is illustrated in Figure 6.2.32.

The operation of creating FLANGE BINDER is similar to BOUNDARY LINE BINDER, with an exception that the user must select either elements or surfaces to define the flanges of a part.

#### DEFINE BINDER ORIENTATION

This function enables the user to define UV-direction of the binder surface. The cross line with arrow controls the surface shape and must be parallel to the direction of the part boundary. Figure 6.2.27 illustrates a typical orientation of the U-V axes relative to the part.

#### SELECT FLANGE ELEMENTS

This function is used to select elements on the flanges of a part. The user selects elements using the functions provided in SELECT ELEMENT dialog box as described in Section 4.3.2.

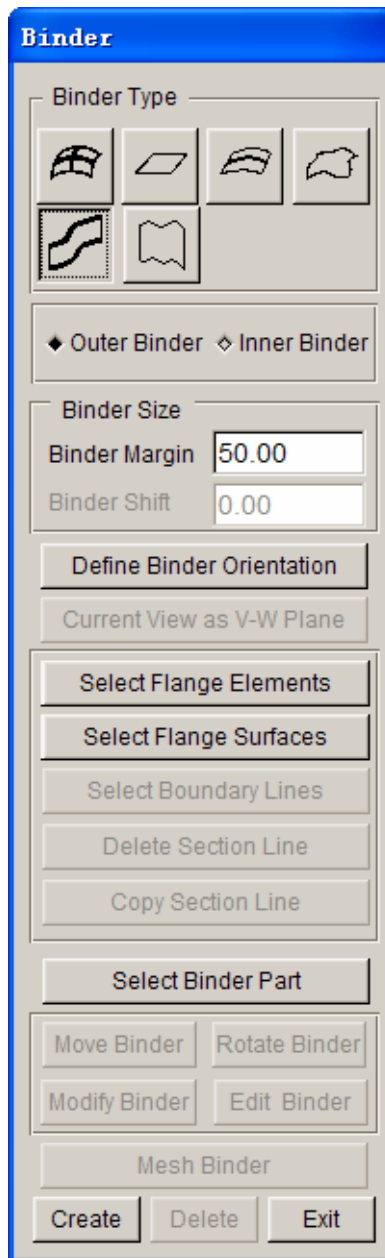


Figure 6.2.32 Flange Binder dialog box

### SELECT FLANGE SURFACES

This function is used to select surfaces on the flanges of a part. The user selects elements using the functions provided in SELECT ELEMENT dialog box as described in Section 4.3.3.

### EDIT BOUNDARY LINES

This function allows the user to edit the boundary lines of the flange surface. Refer to a detailed

description on the EDIT LINE functions in Section 6.2.1.

After the elements or surfaces are selected, click on the CREATE button to create the binder surface. Figure 6.2.33 illustrates typical flange elements on a part. Figure 6.2.34 illustrates a typical binder surface generated using the flange elements as reference.

For a detailed description of other binder editing functions, refer to Section 6.2.1.

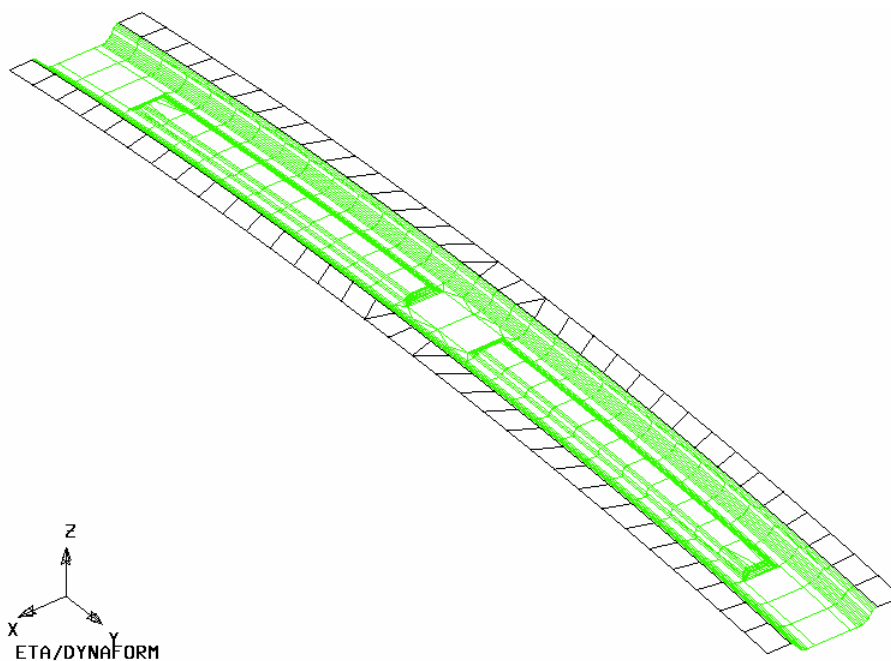


Figure 6.2.33 Selected elements on flanges

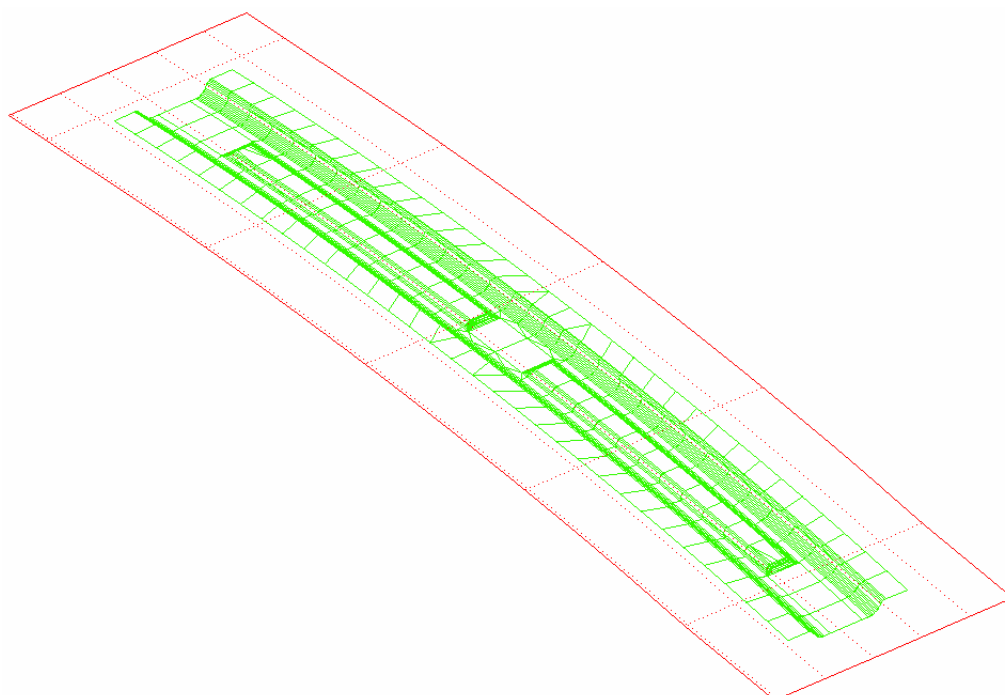


Figure 6.2.34 Typical Flange Binder

## 6.2.6 FREE FORM BINDER

This function is used to generate a binder surface by using the section lines created on the work plane. The dialog box is illustrated in Figure 6.2.35.

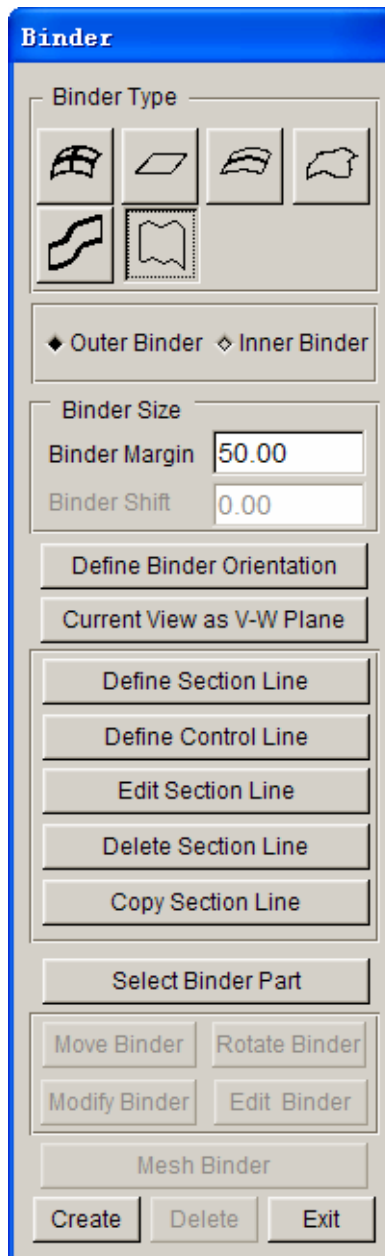


Figure 6.2.35 Free Form Binder dialog box

### DEFINE BINDER ORIENTATION

This function is used to define UV-direction of the binder surface. The line with the arrowhead



controls the surface shape and must be parallel to the direction of the part boundary. Figure 6.2.36 illustrates a typical orientation of the U-V axes relative to the part.

### CURRENT VIEW AS V-W PLANE

This function enables the user to set the current view as workplane. For certain parts, it is not suitable to define the binder U-V axes from the top view using the DEFINE BINDER ORIENTATION function. The user may rotate the part to locate a suitable view for defining section line of the binder, following by clicking on the CURRENT VIEW AS V-W PLANE button to define the binder U-V plane according to the view. eta/DYNAFORM sets the binder V axis parallel to the screen X direction (from left to right). The binder W axis is defined parallel to the screen Y direction (from bottom to top). The binder U axis is obtained by the right hand rule (normal from the screen). In summary, the binder V-W axes are parallel to the screen X-Y axes. The user will be able to define the binder section line in the current view.

### DEFINE SECTION LINE

This function is used to select locations on the screen to define the section line at both ends of the binder along the binder U axis. Click on the DEFINE SECTION LINE button to display the SELECT POINTS dialog box as illustrated in Figure 6.2.36. The part model in binder V-W plane is automatically displayed and the following message is printed in the message prompt window:

***CLICK LOCATIONS ON THE SCREEN TO DEFINE SECTION LINE***

***CLICK LEFT BUTTON TO SELECT, RIGHT TO REJECT, MIDDLE TO EXIT***

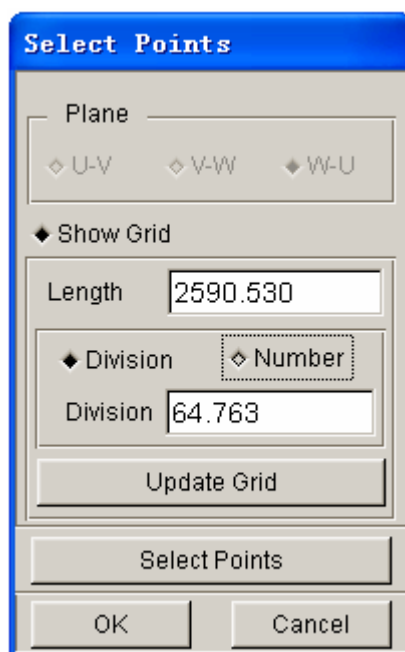


Figure 6.2.36 Selected points dialog box

**SHOW GRID:** Select this option to display the V-W grid plane. Use mouse cursor to select locations on the grid to define the section line.

**LENGTH:** Set the edge length of the grid plane.

**DIVISION:** Set the unit length of the grid plane.

**NUMBER:** Set the number of grid lines.

**UPDATE:** Update the grid plane.

The user may define as many locations on the screen as necessary to define the section line. eta/DYNAFORM extends the section line according to the binder margin defined in the dialog box. Figure 6.2.37 illustrates a typical section line defined on the screen.

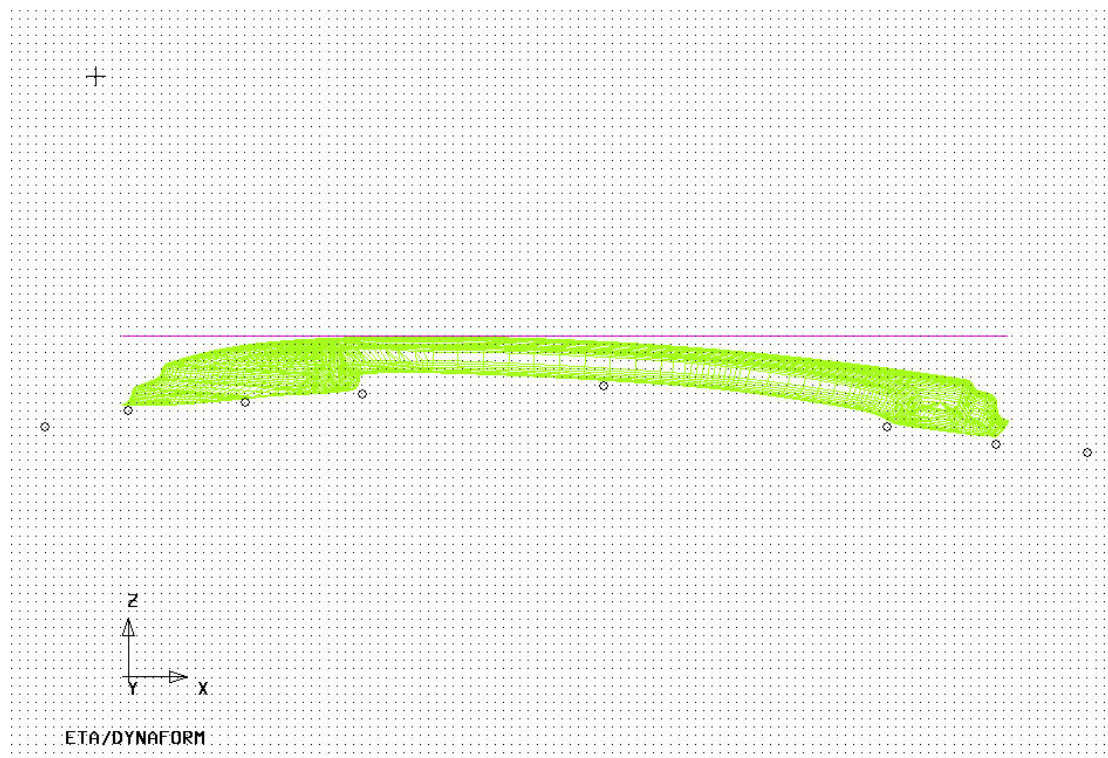


Figure 6.2.37 Typical Section Line

eta/DYNAFORM assigns the defined section line to both ends of the binder (along the binder U direction) according to the binder margin. The user may use the editing function to modify the section either or both section lines.

## DEFINE CONTROL LINE

This function allows the user to define the shape of the binder along the binder U direction. If the

user does not define the control line, straight lines between the section lines at both ends of the binder are automatically created. Once this function is activated, the part model is displayed in binder U-W plane and the following message is printed in the message prompt window.

***CLICK LOCATIONS ON THE SCREEN TO DEFINE CONTROL LINE  
CLICK LEFT BUTTON TO SELECT, RIGHT TO REJECT, MIDDLE TO EXIT***

The user may define as many locations on the screen as necessary to define the control line. eta/DYNAFORM maps the control line to join the section lines. Figure 6.2.38 illustrates a typical control line joining the section lines.

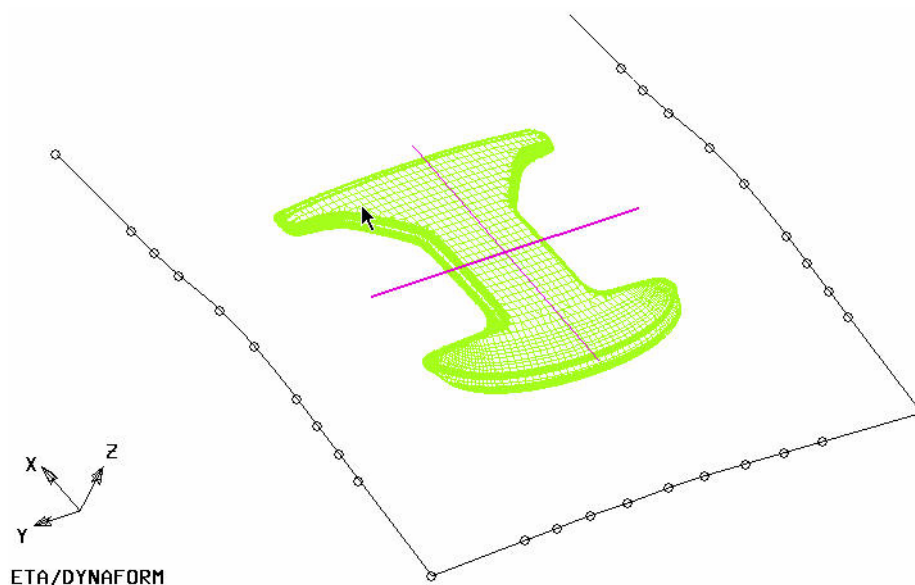


Figure 6.2.38 Typical Control Line Joining Section Lines

### **EDIT SECTION LINE**

This function allows the user to modify the shape of the section line or control line. Please refer to a detailed description in Section 6.2.1.

### **DELETE SECTION LINE**

This function allows the user to delete the section line or control line. If a section line is deleted, eta/DYNAFORM assigns the existing section line to the opposite end of the binder. If the control line is deleted, eta/DYNAFORM returns to the default setting which uses a straight line connecting the section lines at both ends of the binder.

### **COPY SECTION LINE**

This function allows the user to copy the section line to the opposite end of the binder. Figure 6.2.39 illustrates a typical free form binder with a curved control line.

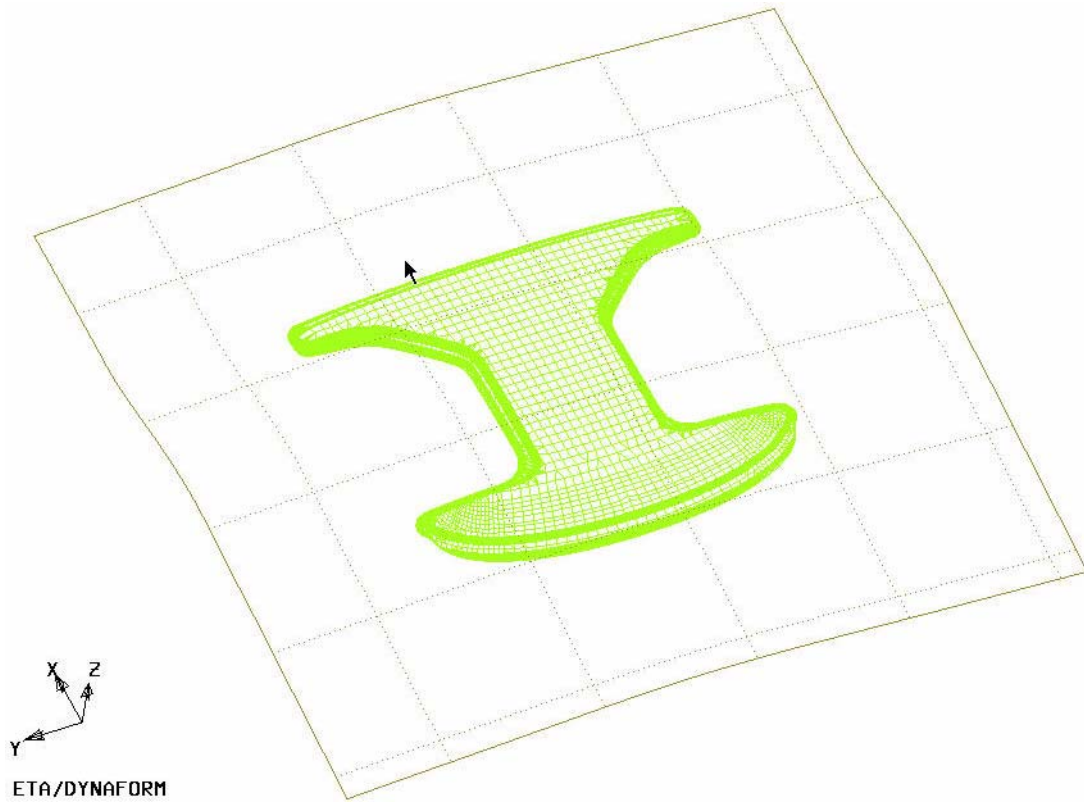


Figure 6.2.39 A Typical Free Form Binder with Control Line

For a detailed description of other binder editing functions, refer to Section 6.2.1.

## 6.3 ADDENDUM DESIGN

This menu provides tools to create and modify addendum surfaces. The ADDENDUM dialog box as illustrated in Figure 6.3.1 contains the functions for the creation and modification of outer and/or inner addenda. The functions in this dialog box are organized into three sections: Master Profiles, Addendum and Profile.

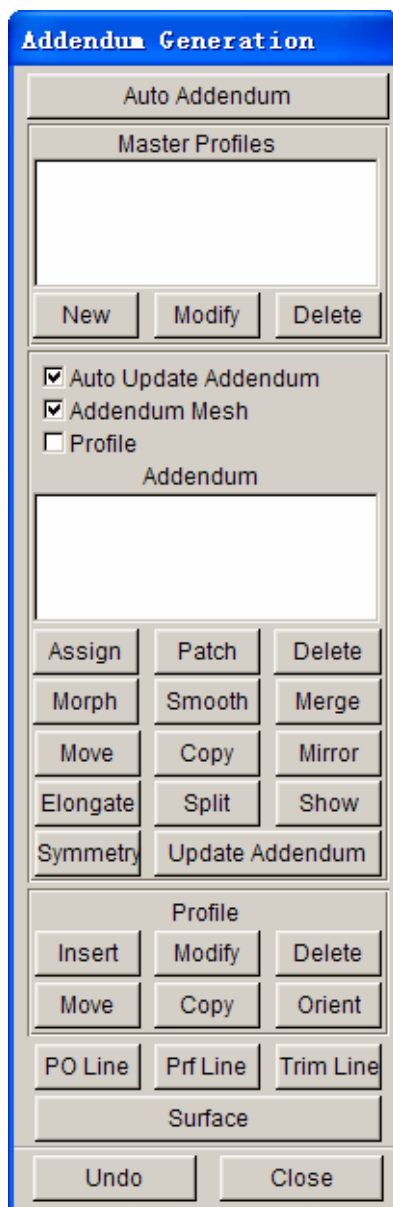


Figure 6.3.1 Addendum Generation dialog box

**NOTE:**

1. *The binder must be defined prior to using the ADDENDUM functions.*
2. *Once the ADDENDUM menu is first initiated, three new parts is automatically created in the database: POP\_LIN, PROFILE and ADDENDUM.*

### 6.3.1 AUTO ADDENDUM

eta/DYNAFORM provides an option to automatically create the addendum. The addendum created is compatible with the manual profile and may be edited by the addendum editing functions. The dialog box is illustrated in Figure 6.3.2.

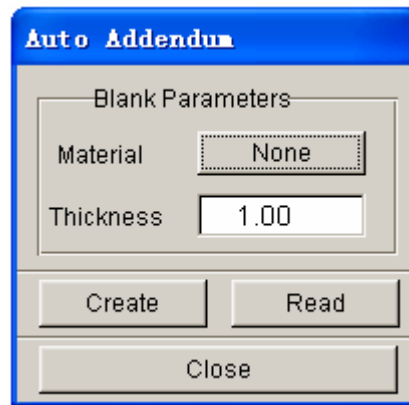


Figure 6.3.2 Auto Addendum dialog box

This function is based on a one-step analysis, therefore, the user is required to provide the material property prior to creating the addendum. Click on the CREATE button to initiate the AUTO ADDENDUM solver to create the addendum. The READ function enables the user to read in a set of profiles which are saved in a \*.txt file.

### 6.3.2 MASTER PROFILE

A profile is defined as cross section line of the addendum that is usually cut along the outer boundary of a part at a direction normal to the boundary. A master profile is used as the primary reference profile for the global addendum geometry. The global setting for profile parameters such as width, height, radii and angles are used as default for other profiles. For various distances between part and binder around the addendum, only height and length may be varied from the master profile.

The displayed master profile does not represent the real geometry at every section on the addendum as the distance between part and binder varies. The displayed master profile is only a schematic drawing with respect to the settings for width, height, radii and angles.

Click on the NEW button from the ADDENDUM GENERATION dialog box to open the PROFILE window. This window shows the master profile with the end section of the part and binder geometry. Figure 6.3.3 illustrates a Type 1 master profile with two radii.

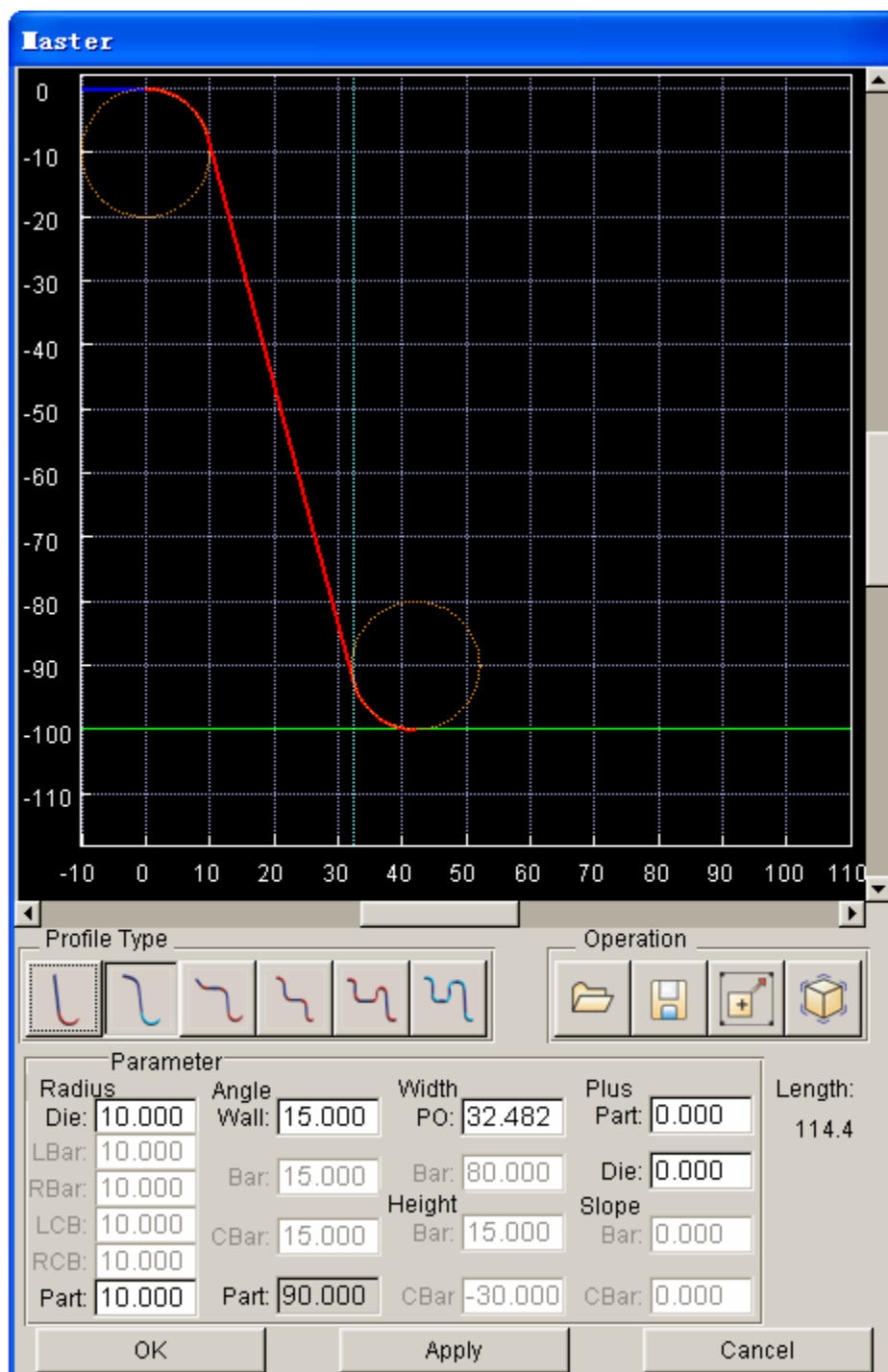


Figure 6.3.3 Master Profile window

The dotted circles represent various radii parameters that can be interactively changed. The user can place mouse cursor near the circle, click and hold left mouse button and drag circle to modify the radius. eta/DYNAFORM dynamically displays the radius in the corresponding input field under the radius column.

For profile as illustrated in Figure 6.3.4, four different dashed lines can be used to adjust the location of the circles. The total length-of-line of the profile is displayed at the lower right corner of the GRAPHIC PROFILE window. The location of lines and size of circles may also be modified from the corresponding input fields.

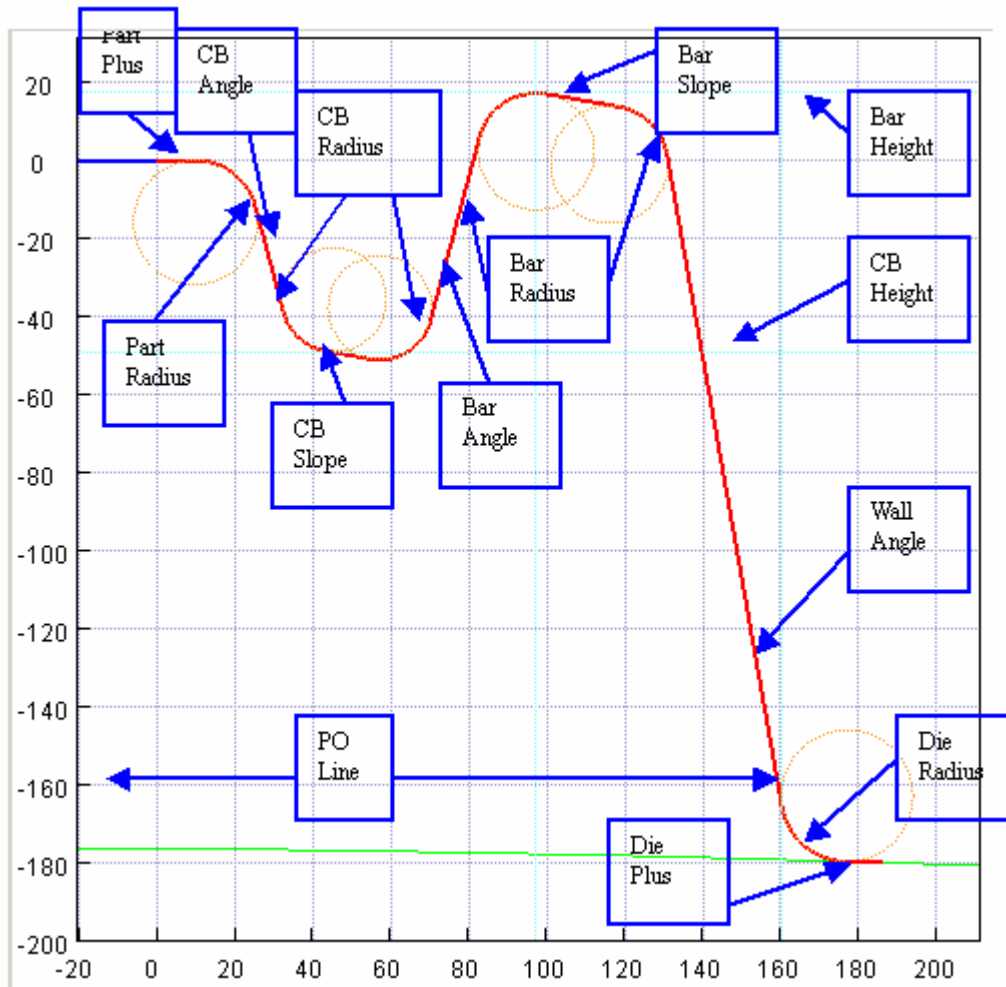


Figure 6.3.4 Various Addendum Parameters

**NOTE:** The user should create different profiles using these lines to further understanding the control parameters. In some cases, some parts of a profile may not be moved in certain directions to prevent unreasonable addendum designs. The user should also change different parameters to check the effects on the profile.

#### WIDTHS:

**BAR:** Determines the width of the drawbar.

**PO Line:** Determines the position of the punch opening line (PO Line).

#### HEIGHTS:

**BAR:** Determines the height of the drawbar.

**CBAR:** Determines the position of the CB height line (Counter Bar) with respect to the part boundary.



There are 6 types of master profiles available in DFE:

### TYPE 1

The TYPE 1 profile is used primarily for parts with vertical edges. This type of profile will produce a tangential transition from the part boundary to the binder surface. As illustrated in Figure 6.3.5, the BAR WIDTH, BAR HEIGHT and CB HEIGHT are disabled in TYPE 1 profile. This type of profile is always extended from the part boundary.

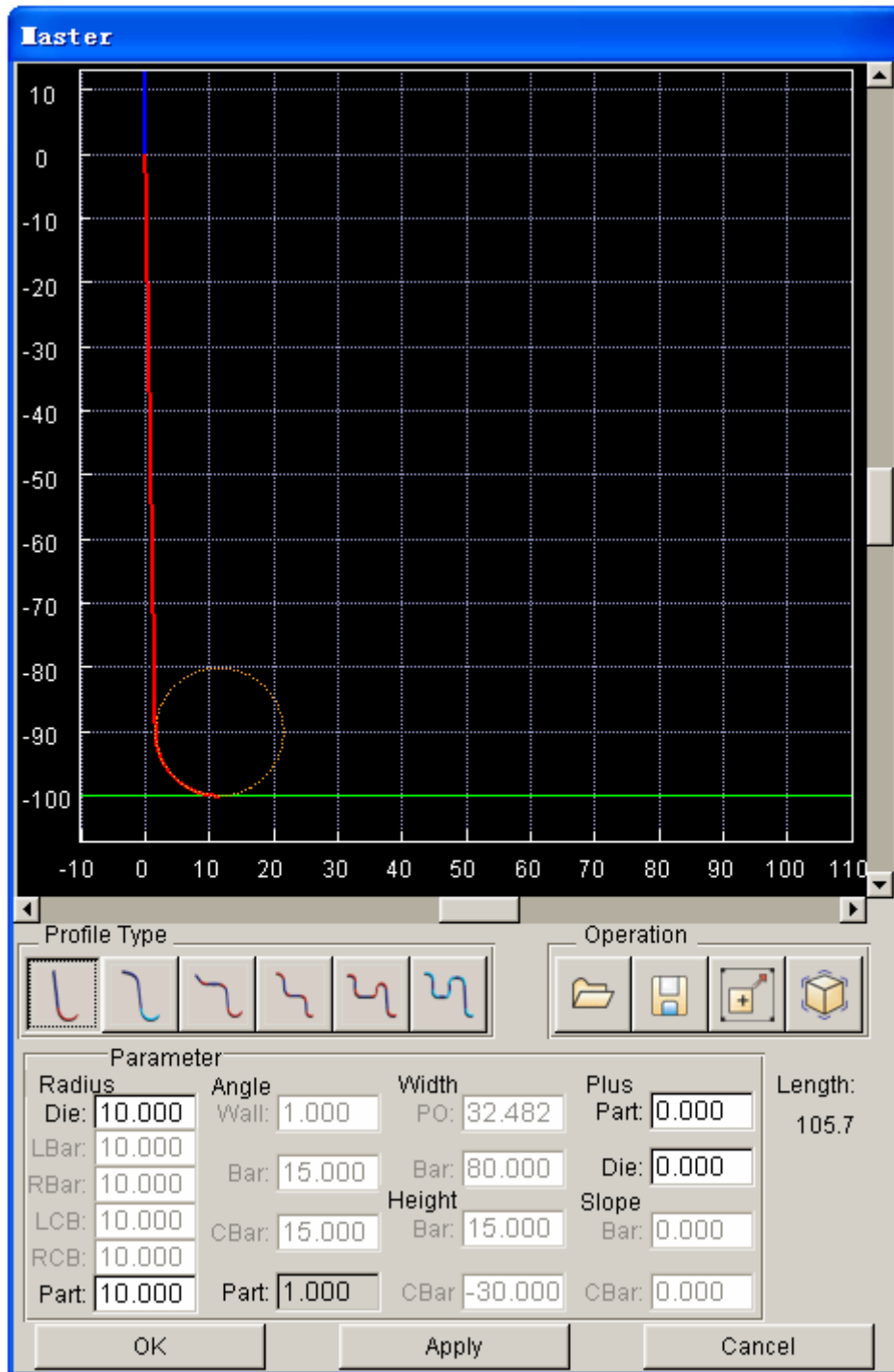


Figure 6.3.5 Profile Type 1 window

**TYPE 2**

The TYPE 2 profile produces the shortest distance between the part boundary and the punch opening line. As illustrated in Figure 6.3.6, the BAR WIDTH, BAR HEIGHT and CB HEIGHT are disabled in TYPE 2 profile.

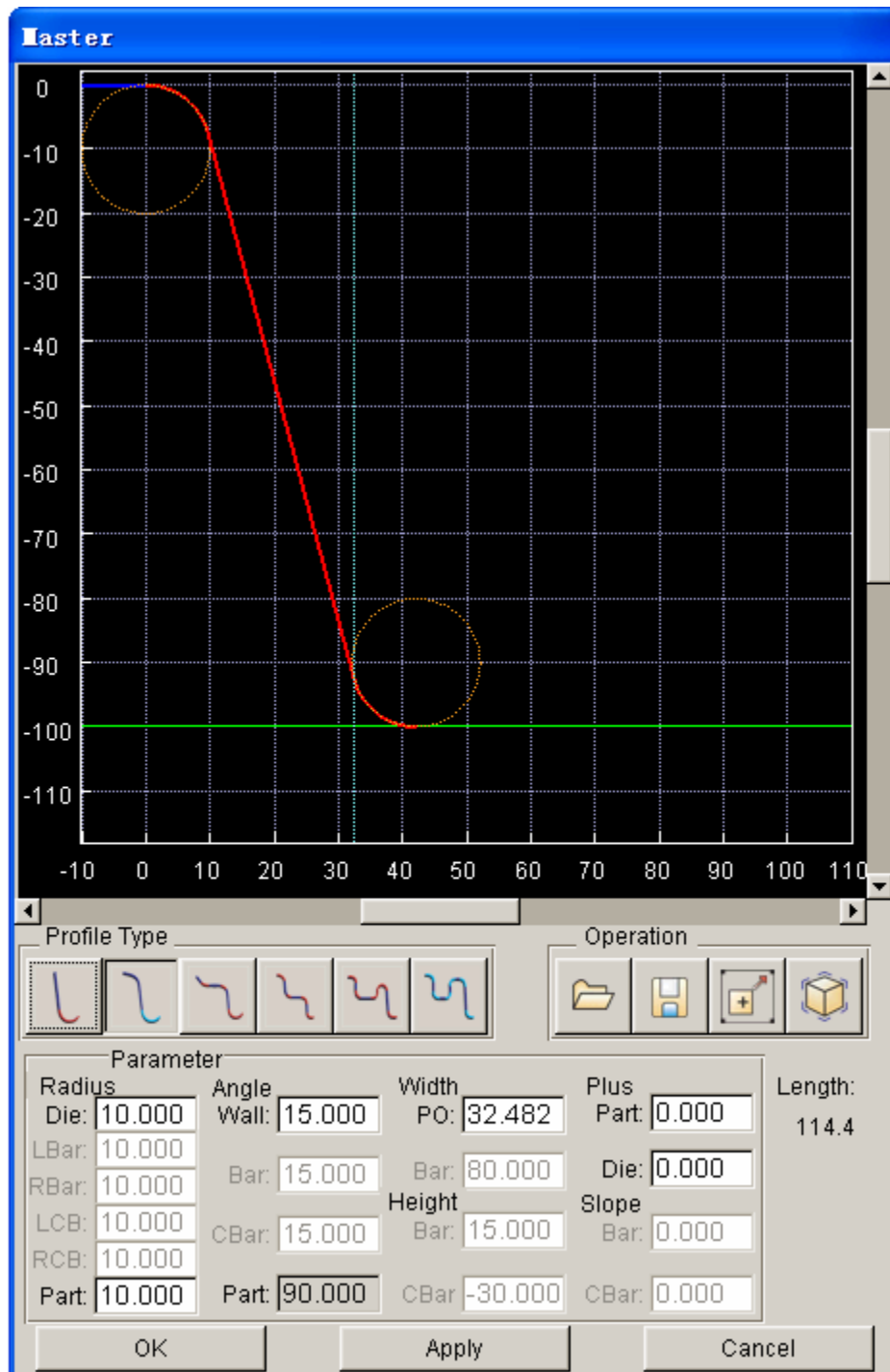


Figure 6.3.6 Profile Type 2 window

The position of the PO width depends on the punch and die radii, and the part plus width (if not zero).

### TYPE 3

The TYPE 3 profile is illustrated in Figure 6.3.7. The height of BAR is measured from the tangential extension of the part. BAR WIDTH is set to minimum and disabled. CB HEIGHT is also disabled.

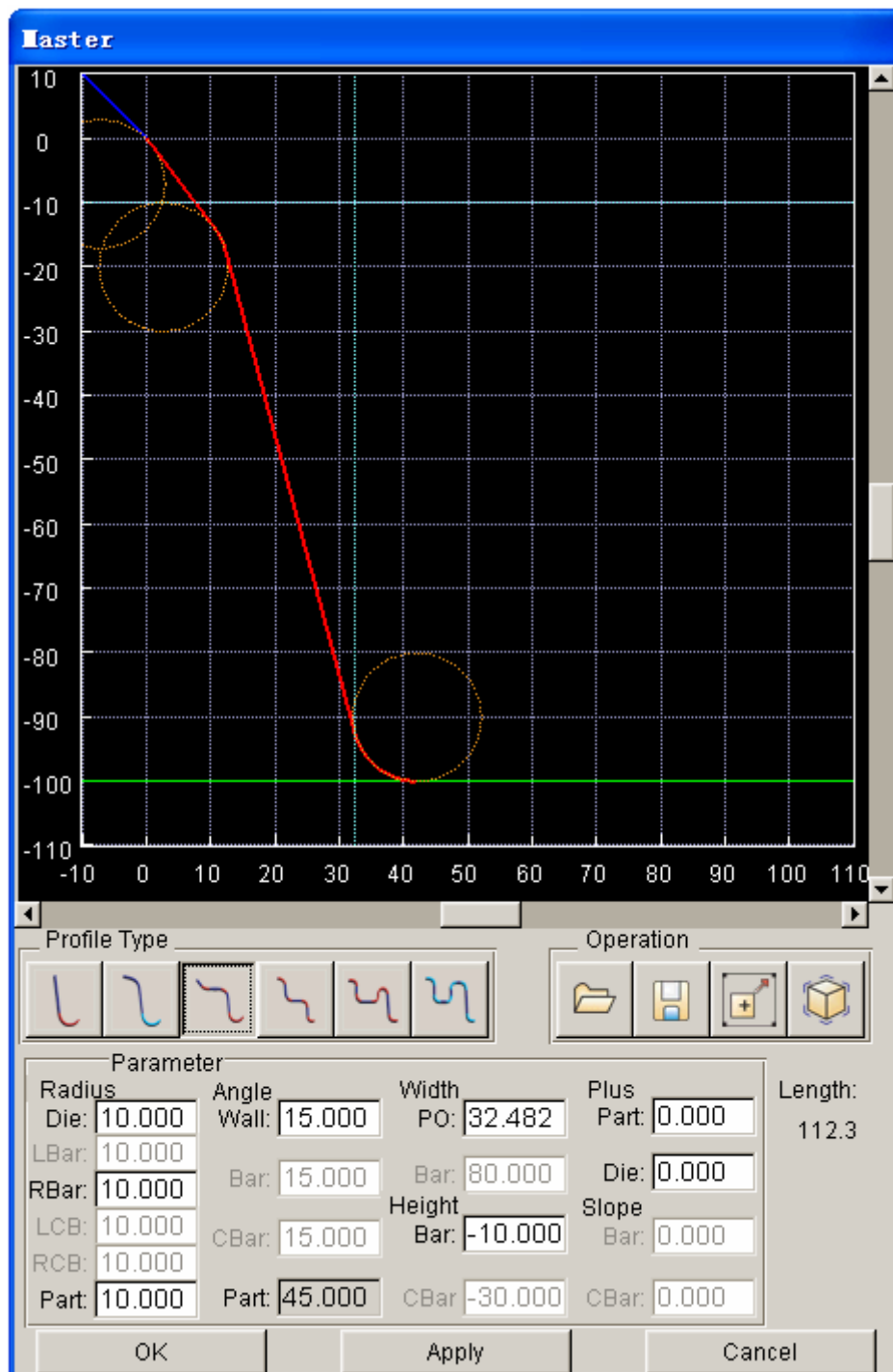


Figure 6.3.7 Profile Type 3 window

**TYPE 4**

The TYPE 4 profile is illustrated in Figure 6.3.8. CB HEIGHT is measured from the part boundary. Both BAR WIDTH and BAR HEIGHT are disabled.

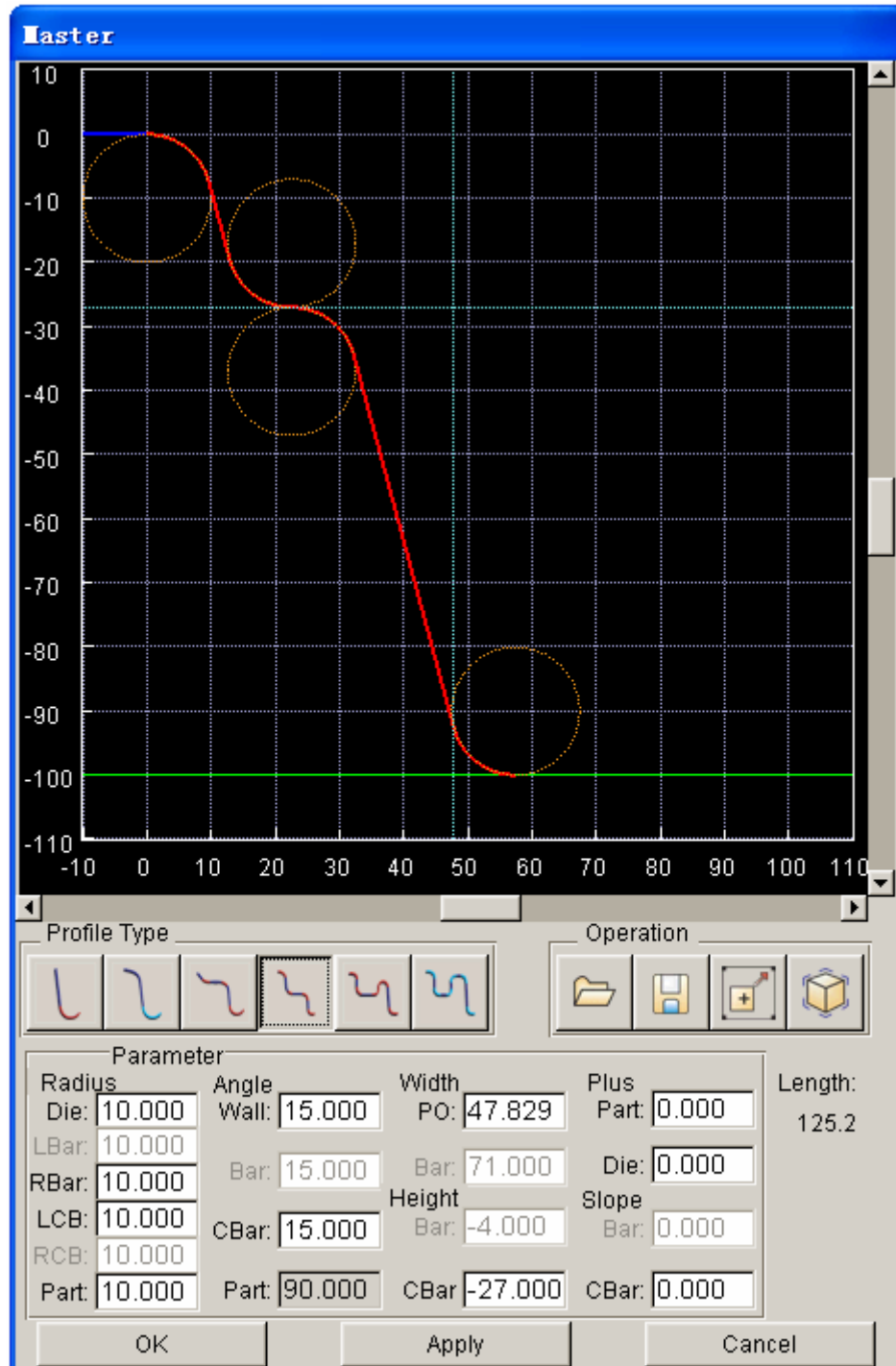


Figure 6.3.8 Profile Type 4 window

**TYPE 5**

The TYPE 5 profile is the combination of TYPE 4 profile with a simple drawbar. As illustrated in Figure 6.3.9, both BAR WIDTH and CB HEIGHT are disabled.

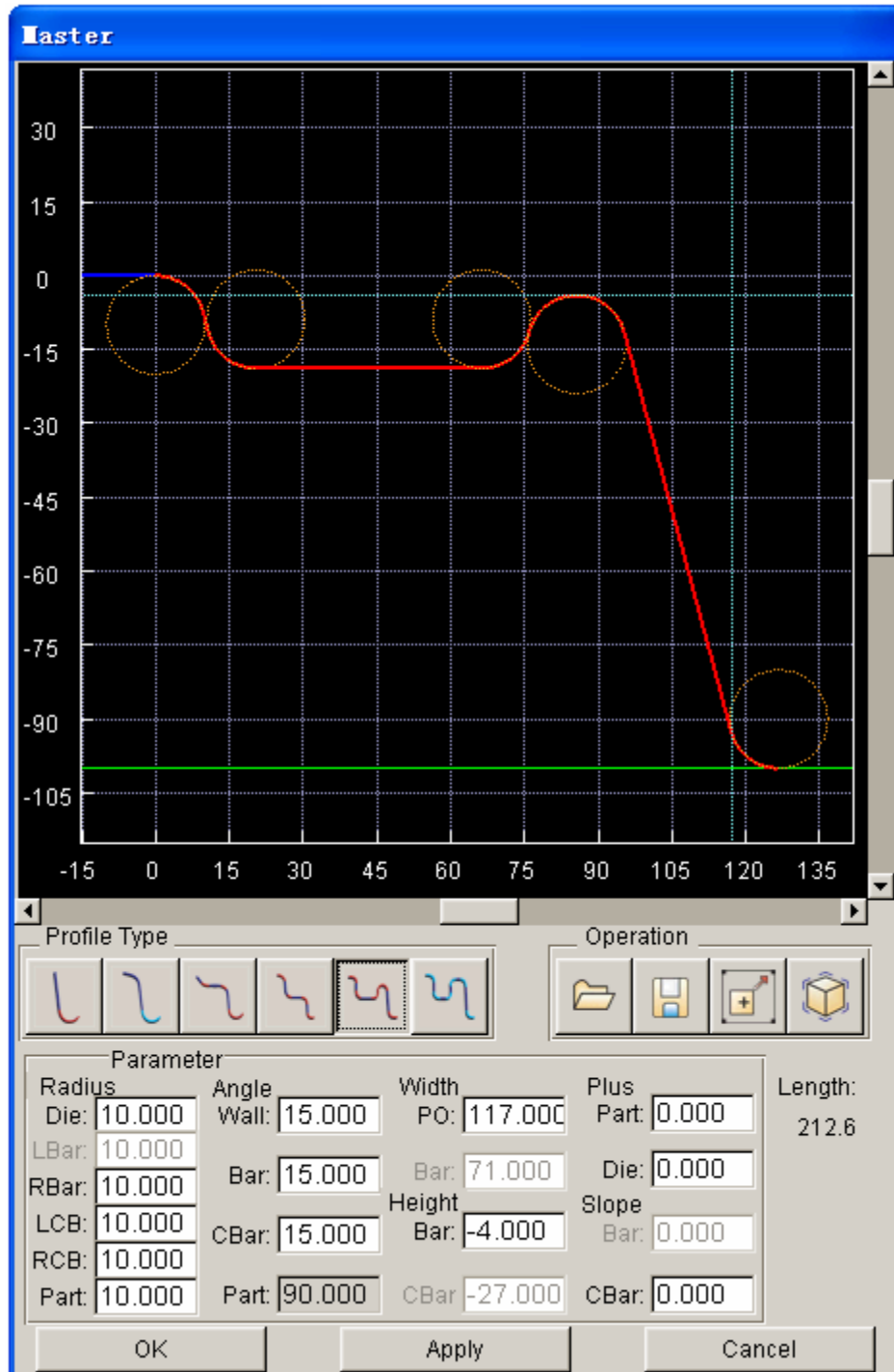


Figure 6.3.9 Profile Type 5 window

**TYPE 6**

The TYPE 6 profile is a combination of TYPE 5 profile with a full-featured drawbar. As illustrated in Figure 6.3.10, all parameters are enabled in this type of profile.

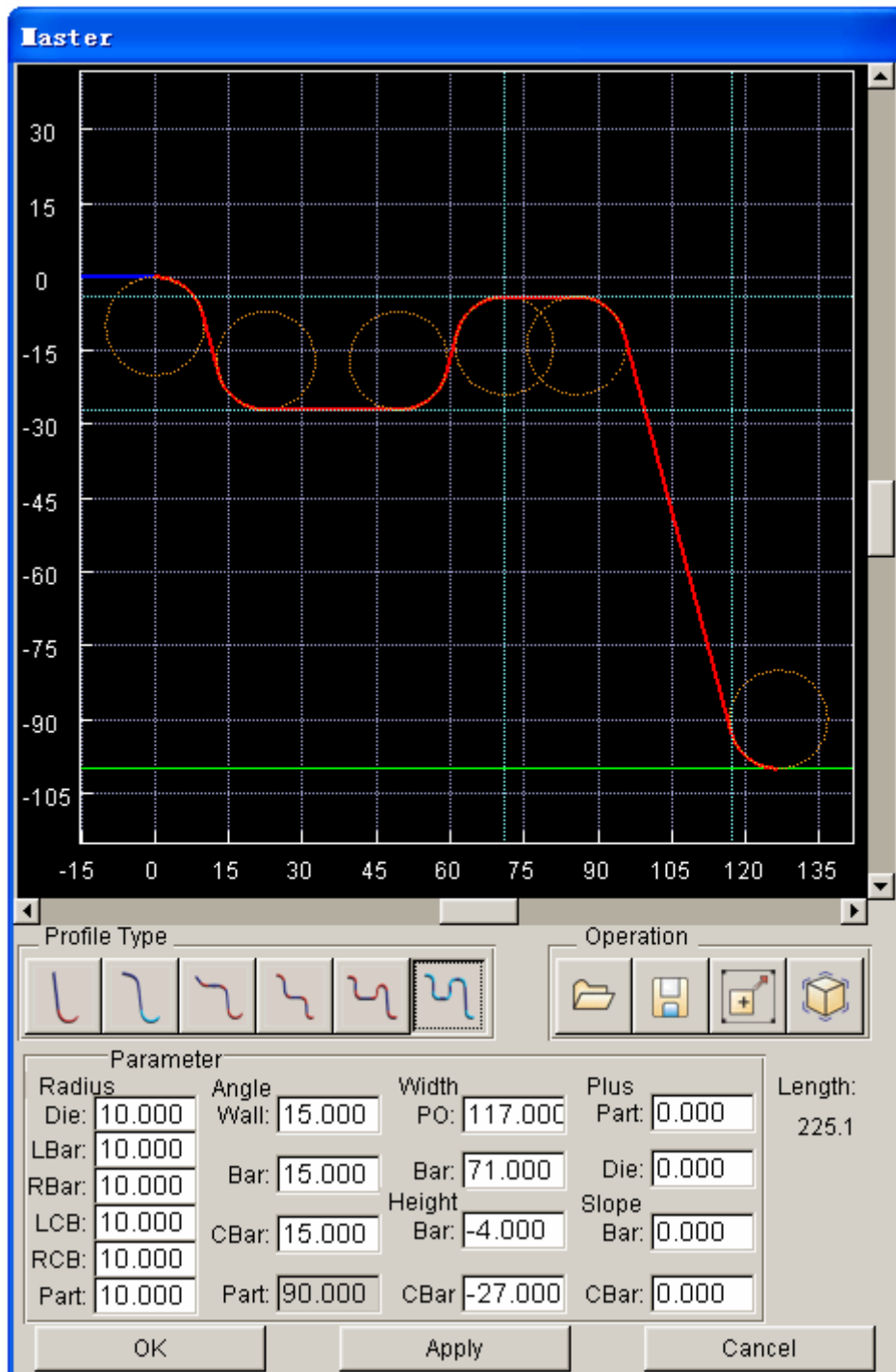
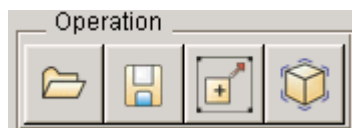


Figure 6.3.10 Profile Type 6 window

**OPERATION**



The functions under the OPERATION field are used to perform input/output and viewing of the master profile.



**SAVE**

This function enables the user to save parameters of the current master profile to a profile (\*.prf) file. Click the SAVE icon to open the dialog window as shown in Figure 6.3.11.

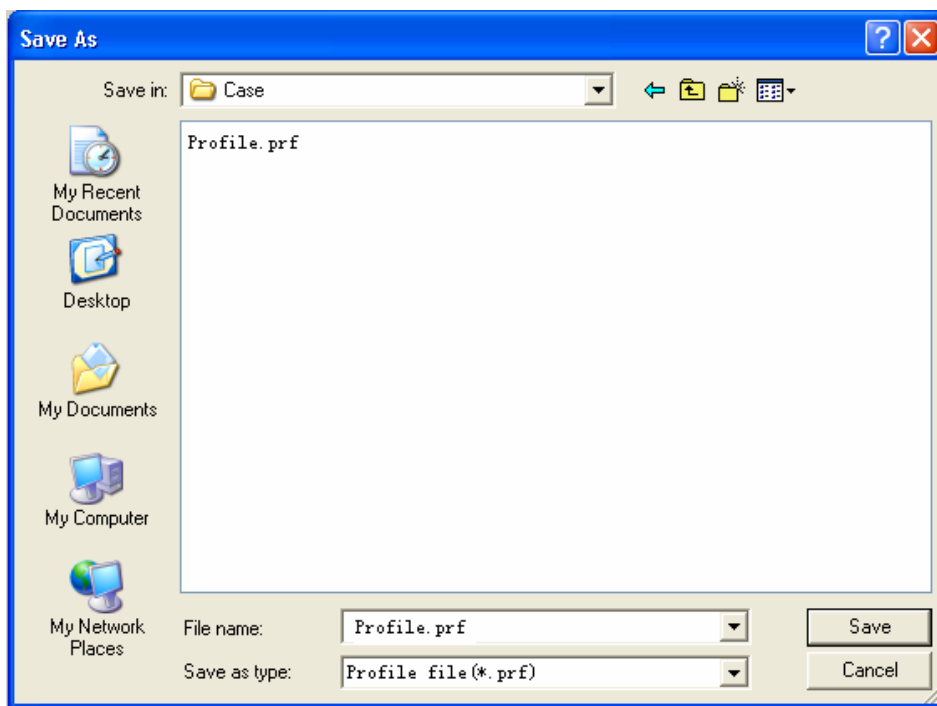


Figure 6.3.11 save Profile Parameters in the File



**OPEN**

This function enables the user to open a profile file (\*.prf) and import its parameters into the current master profile.

**WINDOW ZOOM**

Enable the user to magnify selected area.

**FILL**

Automatically fit the profile to fill in the display window.

After clicking on NEW button to add a master profile, the name of the profile will be listed in the MASTER PROFILE LIST window as illustrated in Figure 6.3.12.

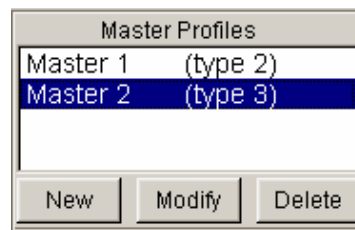


Figure 6.3.12 Master Profile Operations

To modify the master profile, the user selects a master profile from the list, following by clicking on the MODIFY button. The PROFILE window is displayed. The user may modify parameters of the selected master profile in this window.

The user can select a master profile from the list, following by clicking on the DELETE button to remove the selected master profile.

**NOTE:** *eta/DYNAFORM will automatically update the addendum after the master profile is modified. If a segment of the addendum is created based on a master profile, this master profile cannot be deleted until the segment is deleted first.*

### 6.3.3 ADDENDUM

This menu provides functions to create, edit and modify the addendum, etc. The functions provided in the ADDENDUM menu are illustrated in Figure 6.3.13.



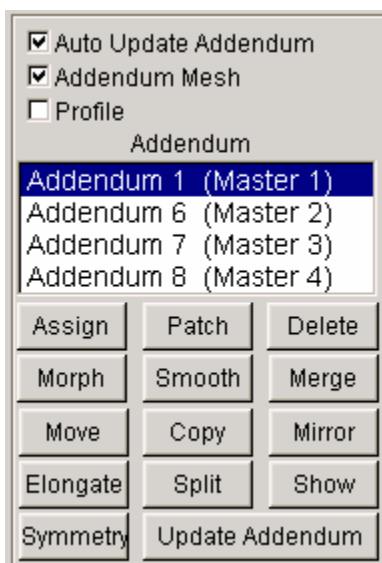


Figure 6.3.13 Create and Edit Addendum

The following three options allow the user to control the update and display of the addendum and profile. Each of these options can be selected individually any time during process of creating addendum.

#### **AUTO UPDATE ADDENDUM (toggle)**

If checkbox of this option is toggled on, the addendum is automatically updated after the profile is modified.

#### **ADDENDUM MESH (toggle)**

If checkbox of this option is toggled on, the addendum mesh is displayed.

#### **PROFILE (toggle)**

If checkbox of this option is toggled on, the profiles are displayed.

#### **6.3.3.1 ASSIGN**

eta/DYNAFORM provides two methods to create the addendum. If checkbox of the BY SEGMENTS option is toggled on, the user is can create the addendum by multiple segments using different master profiles. If the checkbox toggled off, the entire addendum will be created based on a single master profile.

After selecting a master profile, the user clicks on the ASSIGN button to display the dialog box as illustrated in Figure 6.3.14.

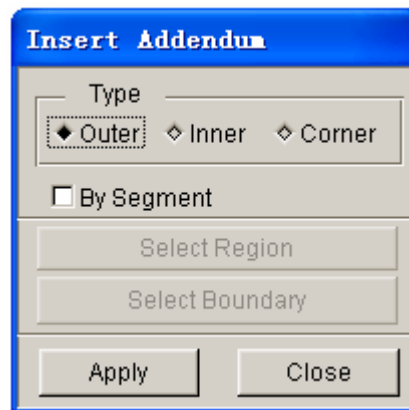


Figure 6.3.14 Insert Addendum dialog box

Select one of the types, OUTER, INNER or CORNER addendum, and click on the APPLY button to insert the addendum.

#### **OUTER ADDENDUM (toggle on OUTER)**

If checkbox of the BY SEGMENTS is toggled on, the SELECT REGION button is enabled. Click on the button to highlight the outer boundary of the part. The following message is printed in the message prompt window:

***SELECT A NODE ON PART BOUNDARY AS STARTING POINT***

After selecting a node on the part boundary, the following message is printed in the message prompt window:

***SELECT A NODE ON PART BOUNDARY AS END POINT***

After two nodes are selected on the outer boundary, click on the APPLY button to create an addendum segment in the defined region. Figure 6.3.15 illustrates a typical addendum segment.

If checkbox of the BY SEGMENT is toggled off, the SELECT REGION button is disabled. The user clicks on the APPLY button to create addendum around the entire outer boundary. Figure 6.3.16 illustrates a typical addendum around the entire out boundary of a part.

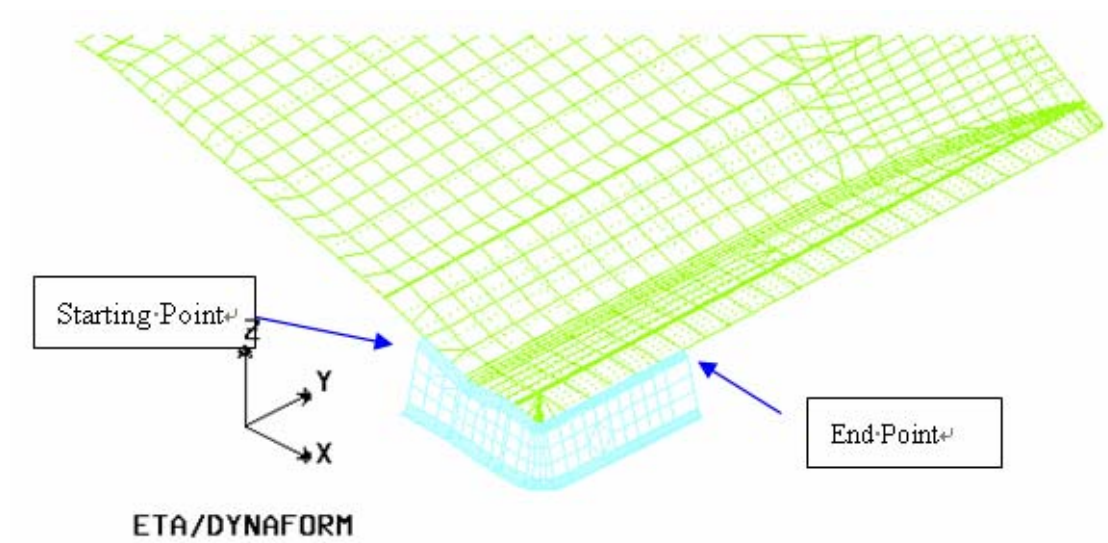


Figure 6.3.15 Create Addendum by Segment

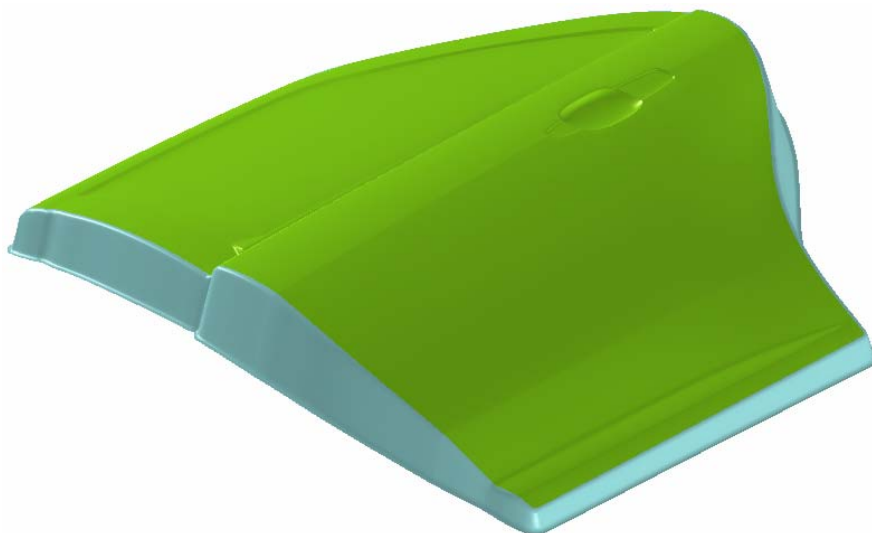


Figure 6.3.16 Complete Addendum by Master Profile

**INNER ADDENDUM (toggle on INNER)**

The SELECT REGION button is enabled if the INNER option is toggled on.

If checkbox of the BY SEGMENTS is toggled on, the user clicks on the SELECT REGION button to select two nodes on the same inner boundary. The user clicks on the APPLY button to generate addendum segment in the selected region.

If checkbox of the BY SEGMENTS is toggled off, the user clicks on the SELECT REGION button to select a node on an inner boundary.

**CORNER ADDENDUM (toggle on CORNER)**

This function is used to create addendum connecting a part with a flange and the neighboring addenda. In this case, the flange may be part of the binder. Refer to Figure 6.3.18 for illustration of a typical part with flange on the binder.

The steps listed below are used to create a corner addendum:

1. When the CORNER option is selected, the SELECT BOUNDARY button as illustrated in Figure 6.3.17 is enabled.
2. Click the SELECT PROFILE button to select a profile on the addendum boundary. The selected profile will be highlighted as illustrated in Figure 6.3.19.
3. Select two nodes on the part boundary as illustrated in Figure 6.3.20. Click on the YES button to accept the defined corner region.
4. Click on the APPLY button to generate the corner addendum as illustrated in Figure 6.3.20.

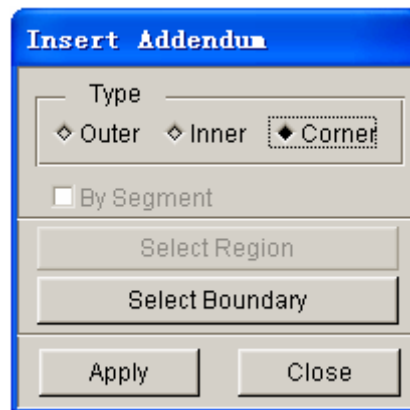


Figure 6.3.17 Corner Addendum

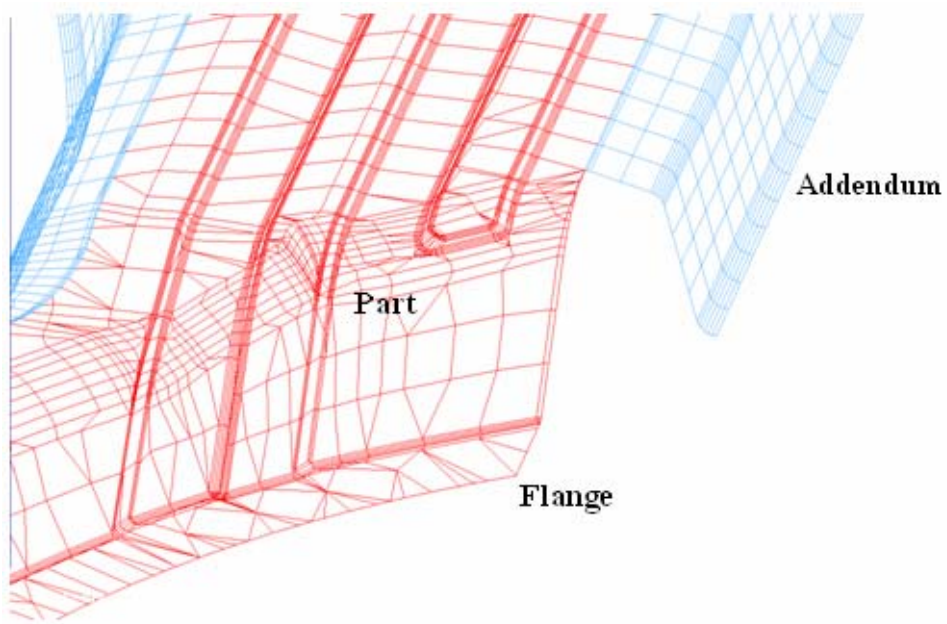


Figure 6.3.18 Flange on Binder Surface

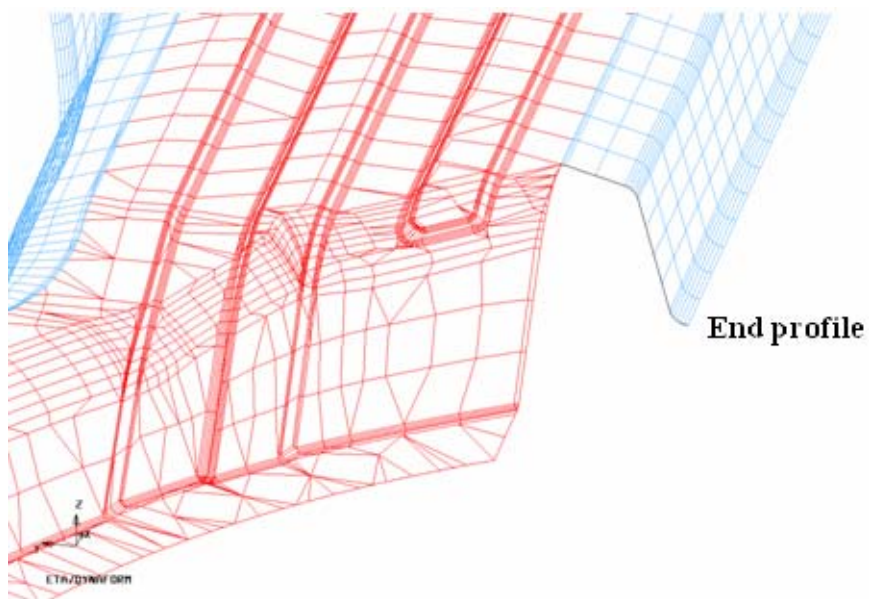


Figure 6.3.19 Select End Profile on Addendum

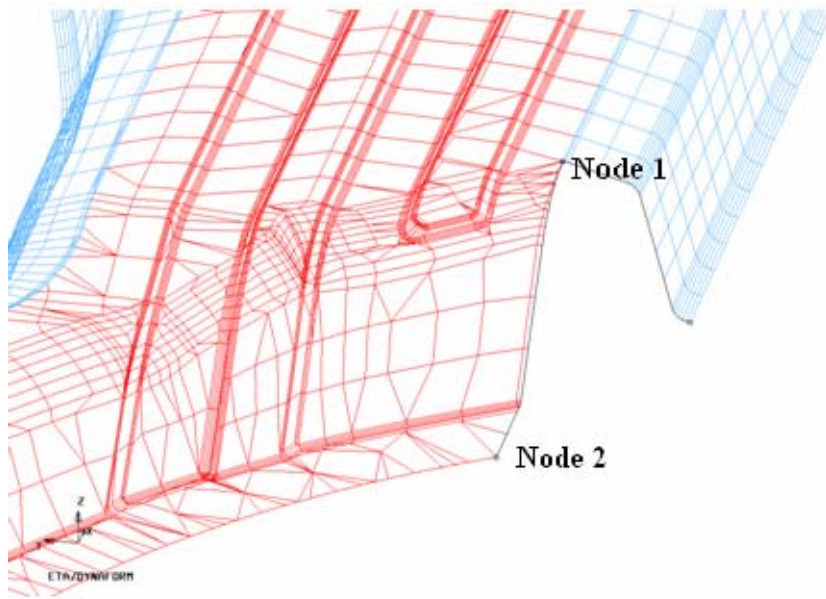


Figure 6.3.20 Select Two Nodes on Part Boundary

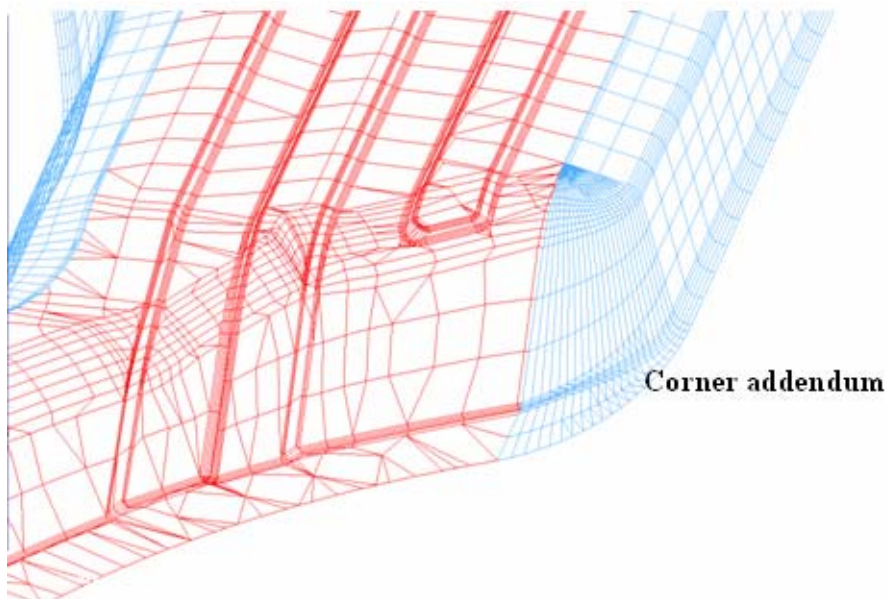


Figure 6.3.21 Generated Corner Addendum

### 6.3.3.2 PATCH

This function is used to fill an addendum segment between two adjacent addenda. The filled segment is typically a transition area between two adjacent addenda generated using different profile types. Figure 6.3.22 illustrates a typical addendum region of different profile types.



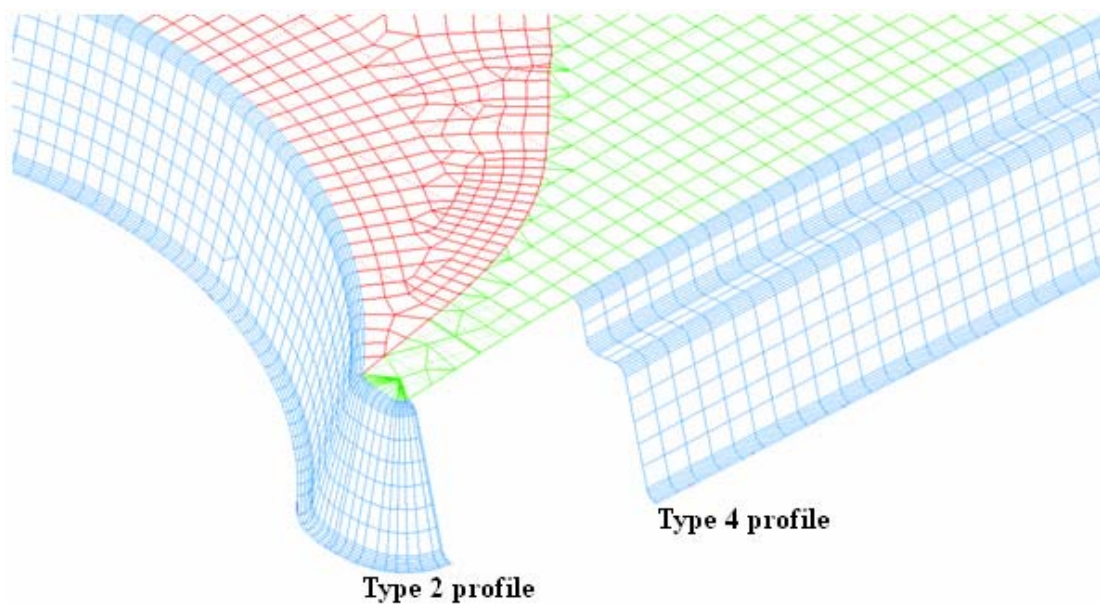


Figure 6.3.22 Transition Area between Adjacent Regions

The user need to select two end profiles of adjacent addenda after clicking on the PATCH button. After the profiles are selected, the segment is created. Figure 6.3.23 illustrates a typical transition area generated by the patch function.

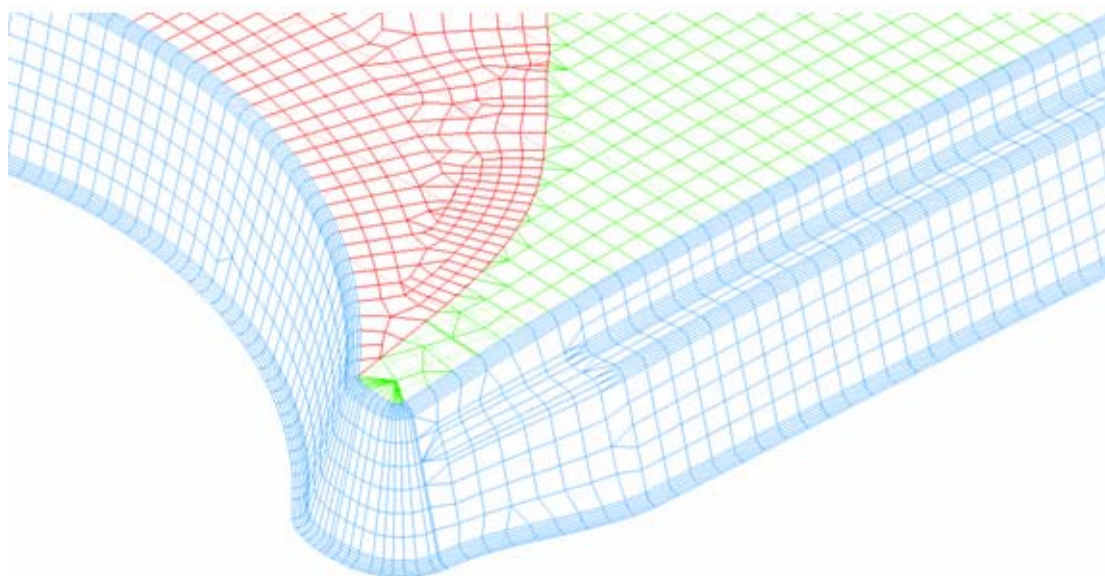


Figure 6.3.23 Typical Result of Patch Function

### 6.3.3.3 DELETE

This function enables the user to remove the selected addendum from database. The user selects an

addendum from the ADDENDUM LIST and clicks on the DELETE button as illustrated in Figure 6.3.13. eta/DYNAFORM highlights the selected addendum and displays a dialog box as illustrated in Figure 6.3.24.

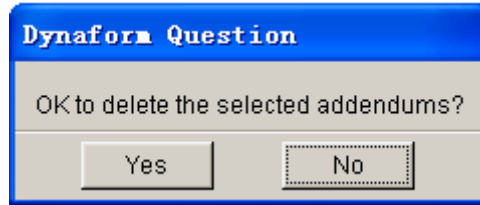


Figure 6.3.24 Dynaform Question dialog box

Click on the YES button to confirm removal of the selected addendum. If NO is selected, the operation is aborted.

#### 6.3.3.4 MORPHING

This function enables the user to modify a segment of addendum by morphing the POP LINE, BAR HEIGHT, BAR WIDTH, CBar HEIGHT and WALL ANGLE. The MORPH ADDENDUM dialog box is illustrated Figure 6.3.25. As illustrated in Figure 6.3.26, eta/DYNAFORM displays the profiles and POP LINE in red and blue, respectively. The UPDATE ADDENDUM function is used to update the addendum after a completion of morphing operation.

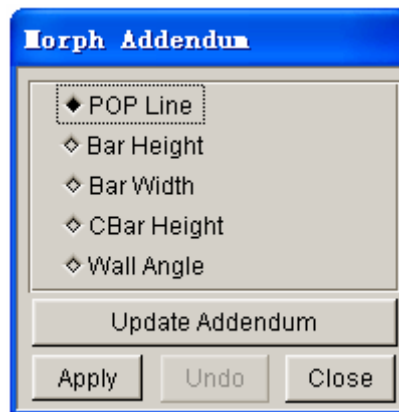


Figure 6.3.25 Morphing Addendum dialog box



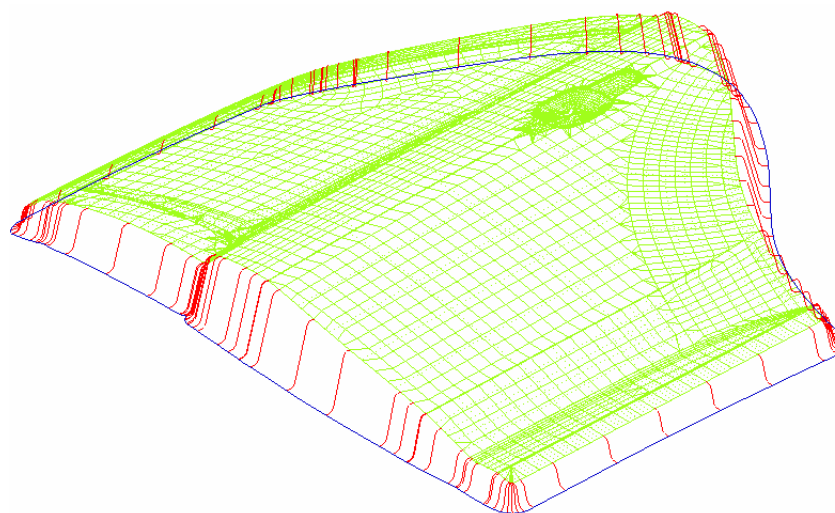


Figure 6.3.26 Profiles and POP Line

### POP LINE

This function allows the user to morph selected region of the POP LINE. The user needs to select the starting and ending profile to define the region which is subjected morphing. If more than one region is defined in the addendum, the starting and ending profiles in each region is highlighted with a circle at each profile. After selecting two profiles, the user needs to confirm the selection in order to highlight the POP LINE as illustrated Figure 6.3.27. The morphing operation of POP LINE is similar to the LINE MORPHING function described in Section 6.4.1.

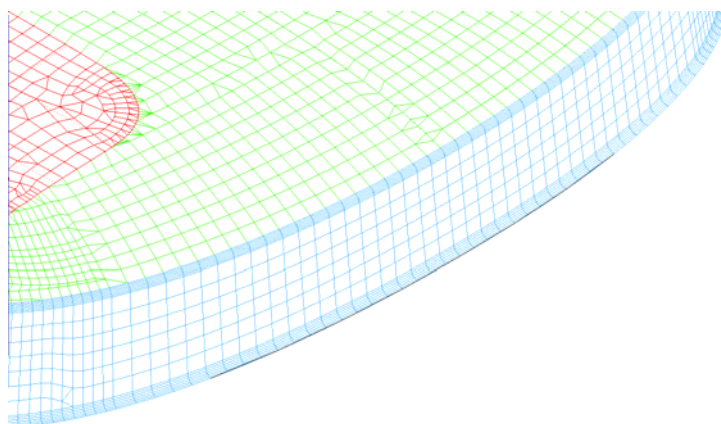


Figure 6.3.27 Selected POP Line for Morphing

Figure 6.3.28 illustrates the morphed POP LINE and the related profiles. After the POP LINE is morphed, the user can click on the CLOSE button to dismiss the dialog box. The profiles and addendum are updated according to the morphed POP LINE. Figure 6.3.29 illustrated the updated

addendum after the POP LINE is morphed.

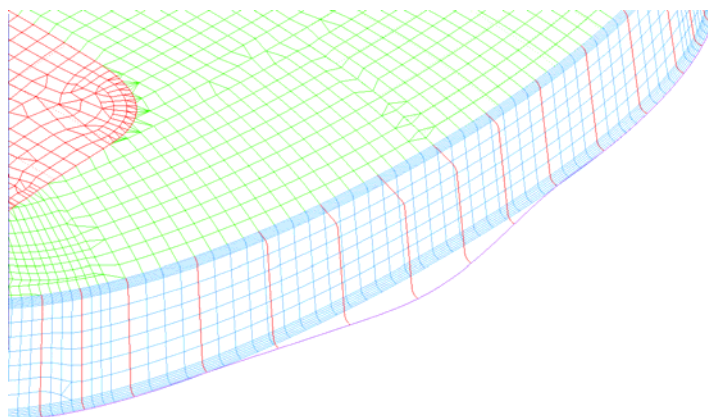


Figure 6.3.28 Morphed POP Line and Profiles

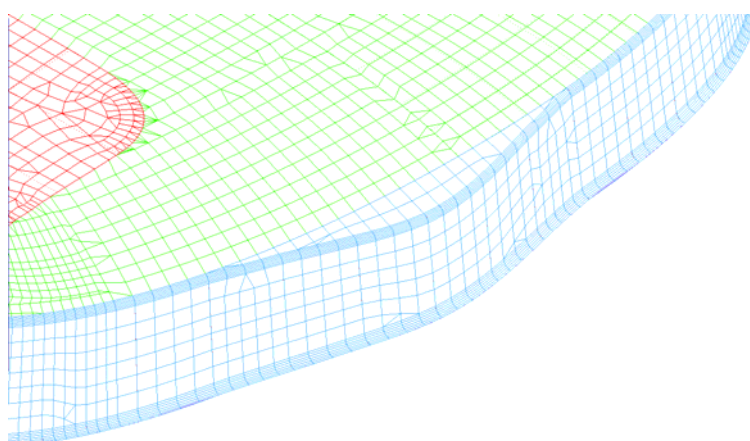


Figure 6.3.29 Updated Addendum According to the Morphed POP Line

### **BAR HEIGHT**

This function allows the user to morph drawbar height of the profiles in the user-defined region. The user selects the starting and ending profile to define the region. The operation is similar to the POP LINE morphing operation, with an exception that the control line on top of the drawbar is morphed to alter the drawbar height of the profiles.

### **BAR WIDTH**

This function allows the user to morph the drawbar width of the profiles in the user-defined region. The user selects the starting and ending profiles to define the region. This operation is similar to the BAR HEIGHT line morphing operation with an exception that the control line on top of the drawbar is morphed to alter the drawbar width of the profiles.

### CBAR HEIGHT

This function allows the user to morph the counter bar height of the profiles in the user-defined region. The user selects the starting and ending profiles to define the region. This operation is similar to the POP LINE morphing operation with an exception that the control line on the bottom of the counter bar is morphed to alter the counter bar height of the profiles.

### WALL ANGLE

This function allows the user to morph the wall angle of the profiles in the user-defined region. The user selects the starting and ending profiles to define the region. This operation is similar to the POP LINE morphing operation with an exception that the POP LINE is morphed to alter the wall angle of the profiles.

#### 6.3.3.5 SMOOTH

This function enables the user to modify a segment of addendum by smoothing the POP LINE, BAR HEIGHT, BAR WIDTH, CBAR HEIGHT and WALL ANGLE. The user selects the SMOOTH function to display a dialog box as illustrated in Figure 6.3.30.

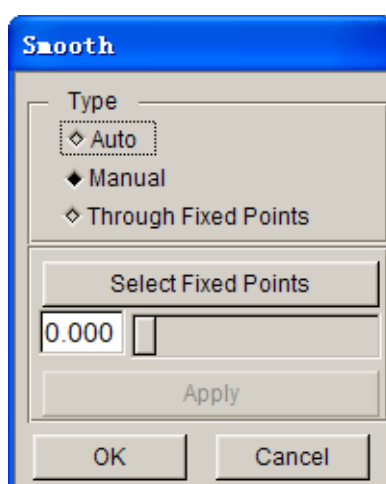


Figure 6.3.30 Smooth dialog box

The options for smoothing operation are quite similar to the morphing function described in previous section. Instead of morphing, the user smoothes the control line.

The following example demonstrates different smooth methods used to smooth the POP LINE. The POP LINE can be smoothed using one of the three options described below. Please refer to Section 6.2.1 for a detailed description of methods used for smoothing.

### **AUTO SMOOTH**

Automatically smooth the entire POP LINE.

### **MANUAL SMOOTH**

Entire POP LINE is smoothed according to the movement of slider. The SELECT FIXED POINTS option enables the user to select more than one fixed point. These selected points will not be moved during the smooth operation. A typical result of manual smoothing with fixed points is illustrated in Figure 6.3.31.

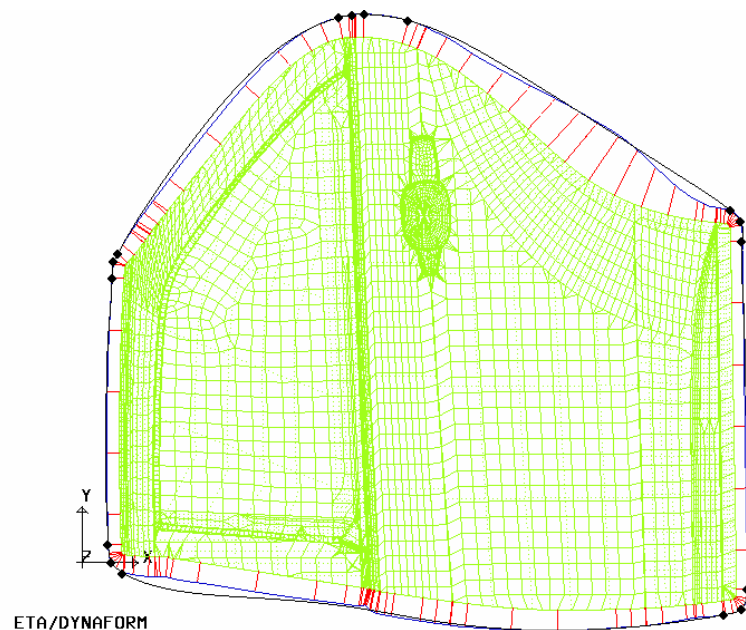


Figure 6.3.31 Manual Smooth With Fixed Point

### **THROUGH FIXED POINTS**

This function allows the user to smooth a segment of the POP LINE by selecting a number of fixed points on the POP LINE. Click on the PREVIEW button to view the smoothed POP LINE. If the smoothed shape is acceptable, clicks on the APPLY to accept the smoothed POP LINE.

Click OK to exit the SMOOTH dialog box. Click on the CLOSE button in the SMOOTH ADDENDUM dialog box to exit the menu. The profiles and the addendum are automatically updated according to the smoothed POP LINE.

### 6.3.3.6 MERGE

This function enables the user to merge (or map) the control line of the POP LINE, BAR HEIGHT, BAR WIDTH, CBAR HEIGHT and WALL ANGLE to a selected line. The profiles are repositioned according to the distance from the control line to the selected line.

#### **POP LINE**

Modifies profiles according to the merged POP LINE.

#### **BAR HEIGHT**

Modifies profiles according to the merged BAR HEIGHT control line.

#### **BAR WIDTH**

Modifies profiles according to the merged BAR WIDTH control line.

#### **CBAR HEIGHT**

Modifies profiles according to the merged CBAR HEIGHT control line.

#### **WALL ANGLE**

Modifies profiles according to the merged WALL ANGLE control line.

Figure 6.3.32a illustrates the definition of a straight line for MERGE POP LINE operation. Figure 6.3.32b illustrates the result of the profiles after the POP LINE is merged to the selected line. The addendum mesh is automatically updated after the MERGE ADDENDUM dialog box is closed.

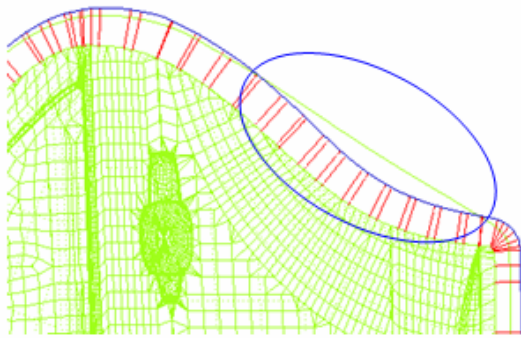


Figure 6.3.32a Merge POP Line

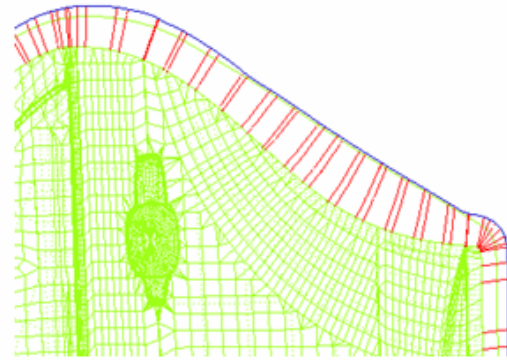


Figure 6.3.32b Merge POP Line

### 6.3.3.7 MOVE

This function enables the user to move a selected addendum segment to another location. If more than one addendum segment are available in the database, the following message is printed in message prompt window:

***PICK A PROFILE GROUP***

After the user selected a group of profile, eta/DYNAFORM highlights the profiles and addendum of the selected group such as illustrated in Figure 6.3.33a. The following message is then printed in the message prompt window.

***PICK NODE ON DIE BOUNDARY AS DESTINATION POSITION***

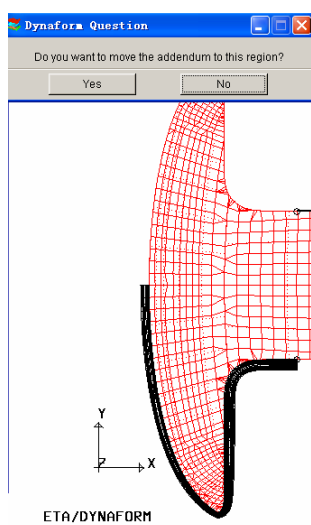


Figure 6.3.33a Move Addendum

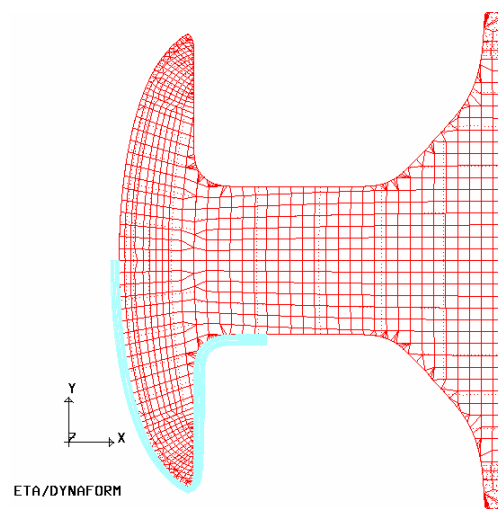


Figure 6.3.33b Move Addendum

A DYNAFORM QUESTION dialog box is displayed requesting the user to confirm the selected location. The new boundary on the part is highlighted, as illustrated in Figure 6.3.33b. If the location of the new boundary is wrong, the user can select NO to put the new boundary on the opposite side of the reference point as illustrated in Figure 6.3.33c. Click on the YES button to accept the boundary location. The selected addendum will be moved to the location as illustrated in Figure 6.3.33d.

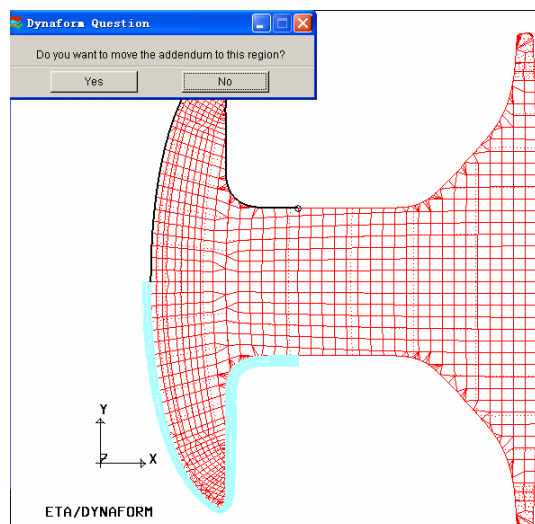


Figure 6.3.33c Move Addendum

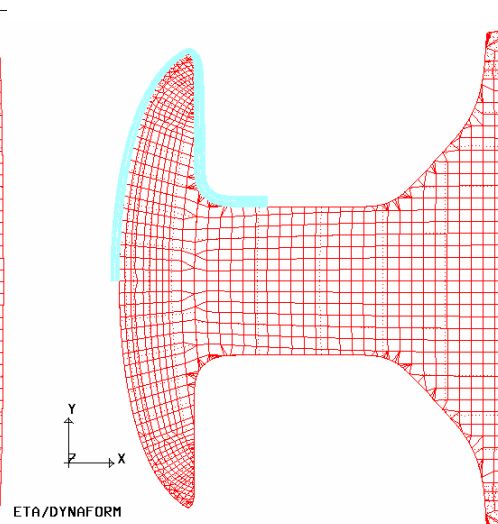


Figure 6.3.33d Move Addendum

### 6.3.3.8 COPY

This function copies the selected profiles and addendum to another location on the part boundary. The operation procedure is similar to the MOVE function with an exception that the original addendum is kept in the database.

### 6.3.3.9 MIRROR

This function generates the entire addendum defined for half symmetry by mirroring it about the symmetry plane. The user must define the symmetry axis for the part in the DFE/PREPARATION menu in order to use this function. Refer to Section 6.1.5 for information about the symmetry definition.

The symmetry axis is displayed after the mirror function is selected. The user selects an addendum segment to display the mirrored boundary. From the displayed DYNAFORM QUESTION dialog box as illustrated in Figure 6.3.34a, the user selects YES to mirror the addendum. The mirrored result is illustrated in Figure 6.3.34b.



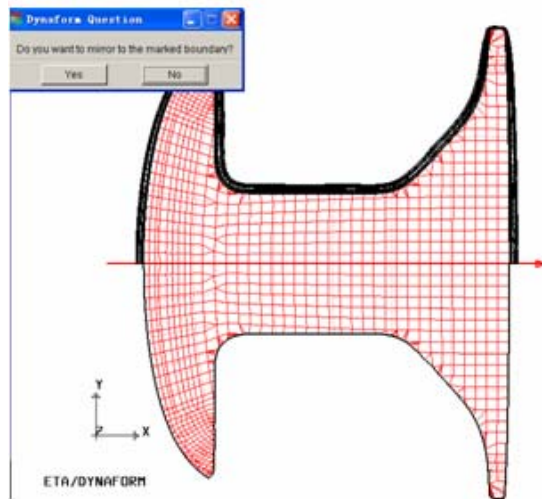


Figure 6.3.34a Mirror Addendum

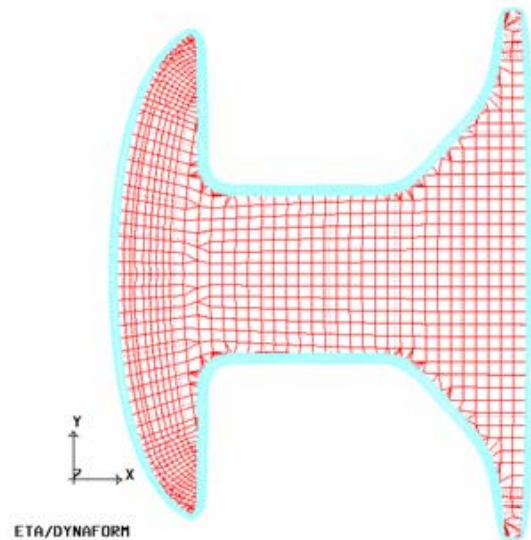


Figure 6.3.34b Mirror Addendum

**NOTE:** If the generated addendum by the MOVE, COPY or MIRROR functions interferes with existing addendum, the operation will be aborted.

### 6.3.3.10 ELONGATE

This function is used to estimate the elongation ratio of the blank along the specified profile direction. It is useful for estimating the level of blank deformation.

If this function is selected, the user needs to select a profile. After selecting a profile, two section lines are highlighted as illustrated in Figure 6.3.35. These section lines are created from the plane of the profile intersecting with the binder and the die.

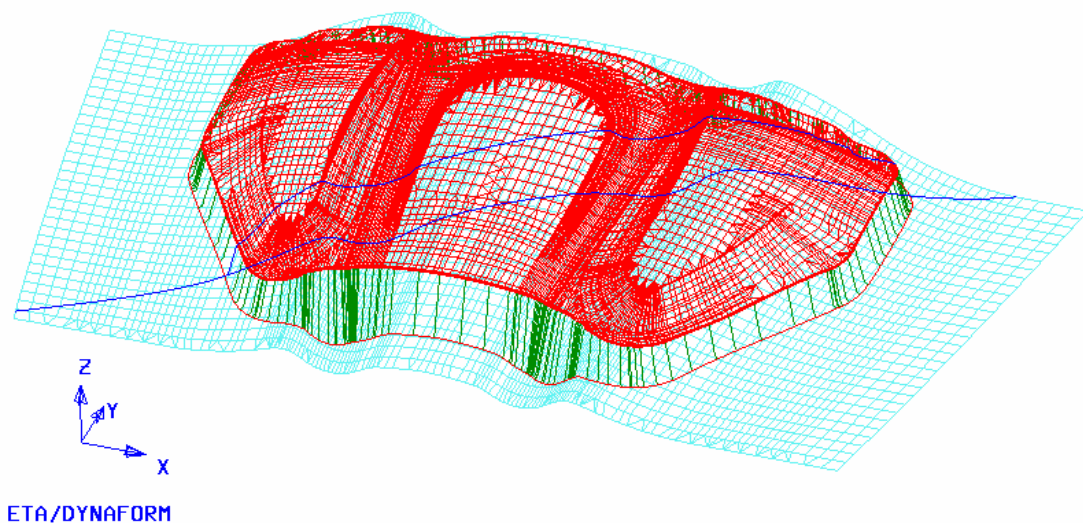


Figure 6.3.35 Select profile



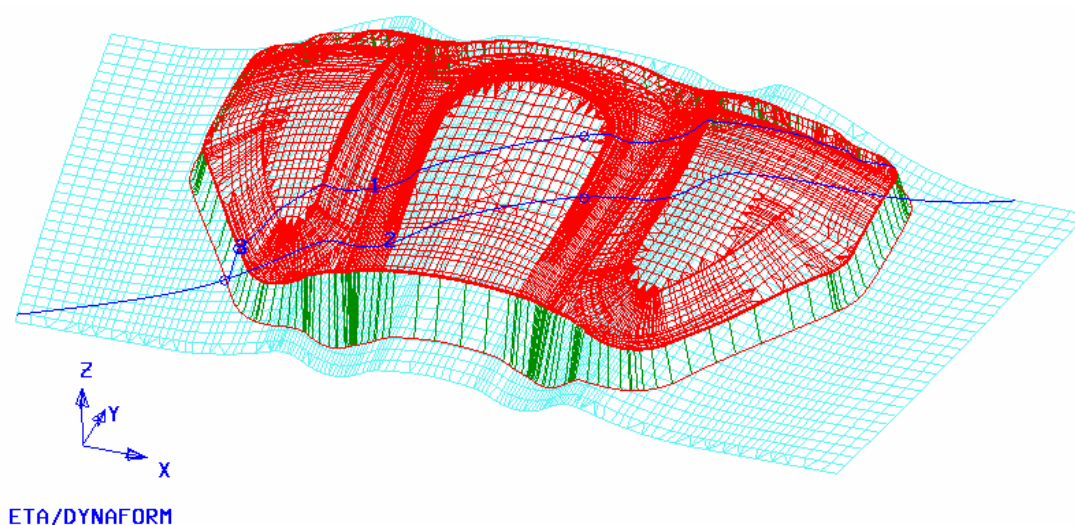


Figure 6.3.36 Point Selection on Line

Next, the user selects a reference point on the highlighted line or clicks the middle button to select the default point. As illustrated in Figure 6.3.36, the lines are divided into three lines identified as 1, 2, 3 respectively. The result is printed in the message prompt window:

***ELONGATION: 5.8%; LENGTH: 1: 434.1 2:521.7 3:207.7***

ELONGATION is the elongation percentage of the specified section line after deformation. It is calculated by the following equation:

$$ELONGATION = (LENGTH\ 1 + LENGTH\ 3 - LENGTH\ 2) / LENGTH\ 2 * 100\%$$

LENGTH 1 and LENGTH 3 indicate the lengths of the punch and addendum after deformation, respectively. LENGTH 2 indicates the length of the line 2 prior to deformation.

Then, the user continues by selecting another profile or click on the right mouse button to exit the function.

### 6.3.3.11 SPLIT

This function enables the user to divide an addendum segment into two independent segments. This function is useful for locally modifying the addendum without changing the shape of other addenda. If only one addendum segment is available in the database, the user can select two different profiles to define the split region. If there is more than one addendum, the user needs to select only one profile in a region. The region is automatically split into two regions with the selected profile. After splitting, a new addendum name is listed in the addendum list, while a master profile type is listed in the master profiles list.

### 6.3.3.12 SHOW

This function helps the user to identify the addendum type with ID number. If the user clicks on the SHOW button, addenda are identified with different ID numbers displayed at the center of each addendum.

### 6.3.3.13 SYMMETRY

This function enables the user to adjust orientation of the profile parallel to the symmetry plane. Prior to mirroring an addendum of a symmetric part, it is necessary to adjust the profiles whose boundary nodes fall in the symmetry plane. If the user selects a profile with a boundary node falls in the symmetry plane, eta/DYNAFORM will automatically change the orientation of the profile to the symmetry plane and requests the user to select other profiles. Figure 6.3.37 illustrates the result of the profiles before and after adjusting the boundary profile.

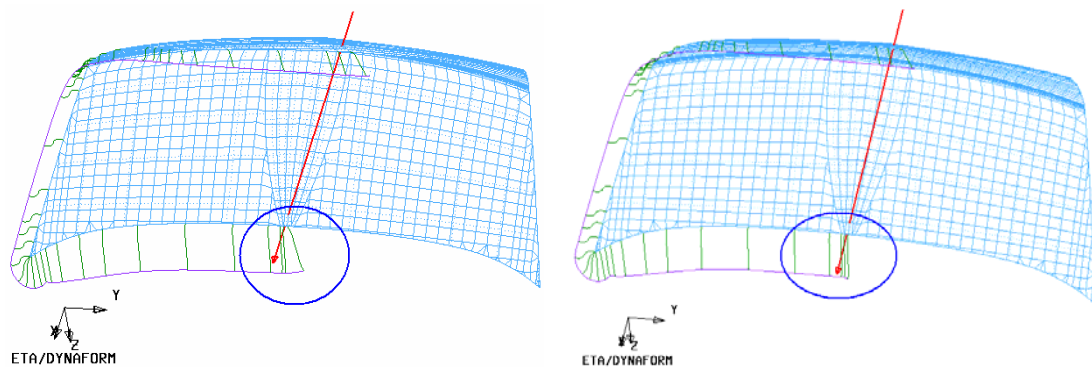


Fig.6.3.367a Before Symmetry is Defined

Fig.6.3.37b After Symmetry is Defined

If the user selects profiles that have no boundary node in the symmetry plane, a DYNIFORM QUESTION dialog box as illustrated in Figure 6.3.38 is displayed. The user clicks on YES button to continue the operation. Eta/DYNAFORM changes the orientation of the selected profile to be parallel to the symmetry plane. The user can click NO to cancel this operation and continues to select another profile.

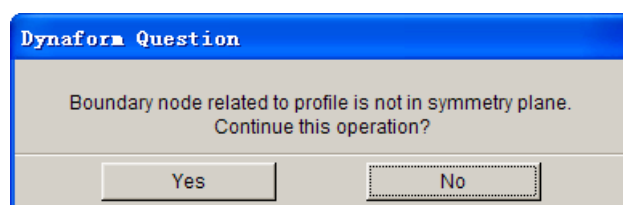


Figure 6.3.38 eta/DYNAFORM Question dialog box

### 6.3.3.14 UPDATE ADDENDUM

If the AUTO UPDATE ADDENDUM option is not selected, eta/DYNAFORM will not update the addendum after the profiles are modified. This function allows the user to update the addendum mesh according to the modified profiles. If the AUTO UPDATE ADDENDUM option is selected, this function is not needed.

## 6.3.4 PROFILE

The profile functions enable the user to INSERT, DELETE, MODIFY or ORIENT one or more profiles. Figure 6.3.39 illustrates the functions provided in the PROFILE section.



Figure 6.3.39 Profile Options

### 6.3.4.1 INSERT

This function enables the user to insert a profile between two adjacent profiles of the same profile group. The profiles and POP LINE are displayed and the following message is printed in the message prompt window:

***SELECT FIRST PROFILE***

After the first profile is selected, the following message is printed in the message prompt window:

***SELECT SCEOND PROFILE***

After two adjacent profiles are selected, the following message is printed in the message prompt window:

***SELECT NODE ON PART AS STARTING POINT***

After the user selects a node on the part boundary, the following message is printed in the message prompt window:

***SELECT A POINT ON POP LINE***

After a point on the POP LINE is selected, a profile is inserted between the boundary node and the point on the POP LINE. The profile type of the inserted profile is similar to the original profile.

### 6.3.4.2 DELETE

This function enables the user to remove one or more profiles. The dialog box is illustrated in Figure 6.3.40.

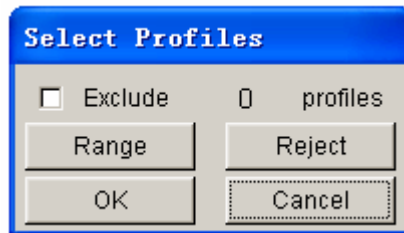


Figure 6.3.40 Select Profiles dialog box

The DEFAULT option allows the user to select only a single profile by using the cursor.

#### **RANGE**

If RANGE is selected, the following message is printed in the message prompt window:

***SELECT STARTING PROFILE***

After starting profile is selected, the following message is printed in the message prompt window:

***SELECT END PROFILE***

After the end profile is selected, the profiles between the starting and ending profiles are highlighted. The user continues the operation by selecting OK to remove the highlighted profiles.

#### **REJECT**

This function enables the user to reject the selected profile.

#### **OK**

This function deletes the selected profiles.

#### **CANCEL**

This function exits the operation without deleting any profiles.

#### **EXCLUDE**

If this option is selected, the selected profile will be excluded from the previous selection.

### 6.3.4.3 MODIFY

This function enables the user to modify a profile and automatically adjust the adjacent profiles to obtain a smooth POP LINE.

- Select two profiles to define the region.
- Select a profile with the cursor, or use the RANGE option to select a group of profiles in this region.
- After the profiles are selected, the PROFILE window is displayed. The user can modify profile types and/or profile parameters. As profiles are modified, eta/DYNAFORM will update the profile in the graphic display window.

eta/DYNAFORM will replace the selected profiles with the modified profile and update other profiles in the selected region to maintain a smooth POP LINE. If the AUTO UPDATE ADDENDUM option is selected, the addendum mesh is automatically updated according to the new profiles.

### 6.3.4.4 ORIENTATION

The addendum consists of a number of profiles between the part boundary and the POP LINE. This function allows the user to change the orientation of a selected profile relatively to the POP LINE.

If the function is selected, the following message is printed in the message prompt window:

***SELECT PROFILE TO ORIENTATION***

After a profile is selected, the user needs to pick a location on the POP LINE. The selected profile is moved to a plane defined by the original part boundary node and the new POP LINE location. After completing the profile orientation operation, the user clicks on the right mouse button to exit the function. The addendum mesh is automatically updated if the AUTO UPDATE ADDENDUM option is selected.

### 6.3.4.5 COPY

This function enables the user to copy an existing profile to another location. After selecting a profile, the user needs to select a node on the boundary of part as the starting point, following by selecting a point on the POP LINE as the second point. After locating the two positions, the profile is copied to the new location.

### 6.3.4.6 MOVE

This function enables the user to move existing profiles to another location.

*NOTE: The two end profiles defining an addendum region cannot be moved.*

### 6.3.5 CREATE PO LINE

This function enables the user to create a PO (Punch Opening) LINE in the current part. The created lines are stored in the part named ADDENDUM.

### 6.3.6 CREATE PRF LINE

This function enables the user to create geometric lines based on the profiles. The user can export these lines as CAD data for further modification. The created lines are stored in the part named ADDENDUM.

### 6.3.7 TRIM LINE

This is a new function added in the eta/DYNAFORM 5.5 DFE module. This function enables the user to determine the trim line of the flange of the part. The trim line is determined so that the sheet boundary takes the desired geometry at the end of the forming steps, following by the trimming operation. The dialog box of TRIM LINE is illustrated in Figure 6.3.41.

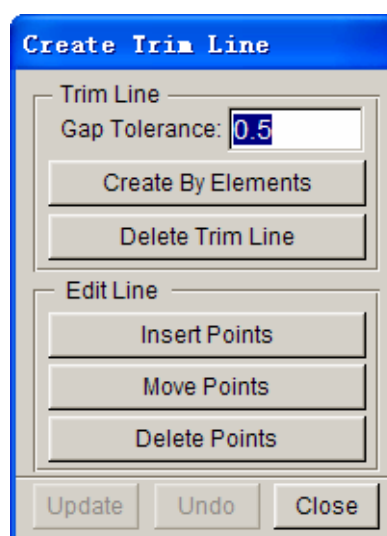


Figure 6.3.41 Create Trim Line dialog box

### GAP TOLERANCE

This parameter sets the tolerance between the selected flange and the addendum elements. If the gap between the flange and the addendum is larger than the defined tolerance, the flange boundary is selected as the outer boundary.

### CREATE BY ELEMENTS

This function enables the user to select the flange elements which is used to obtain the trim line on the addendum. The user selects elements using the functions provided in the displayed SELECT ELEMENTS dialog box. The trim line is automatically created on the addendum or the binder surface. Please refer to Figure 6.3.42 and Figure 6.3.43 for illustration of CREATE TRIM LINE operation.

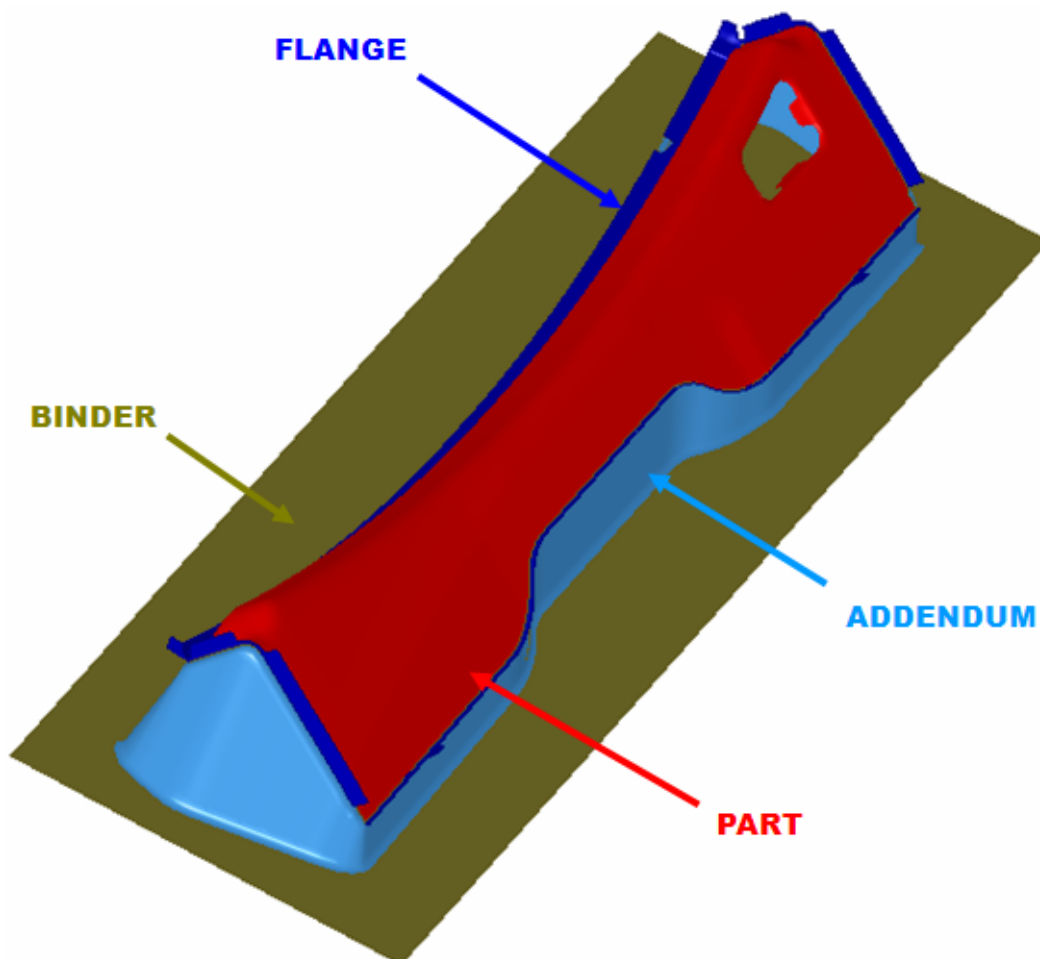


Figure 6.3.42 Before Create the Trim Line operation.

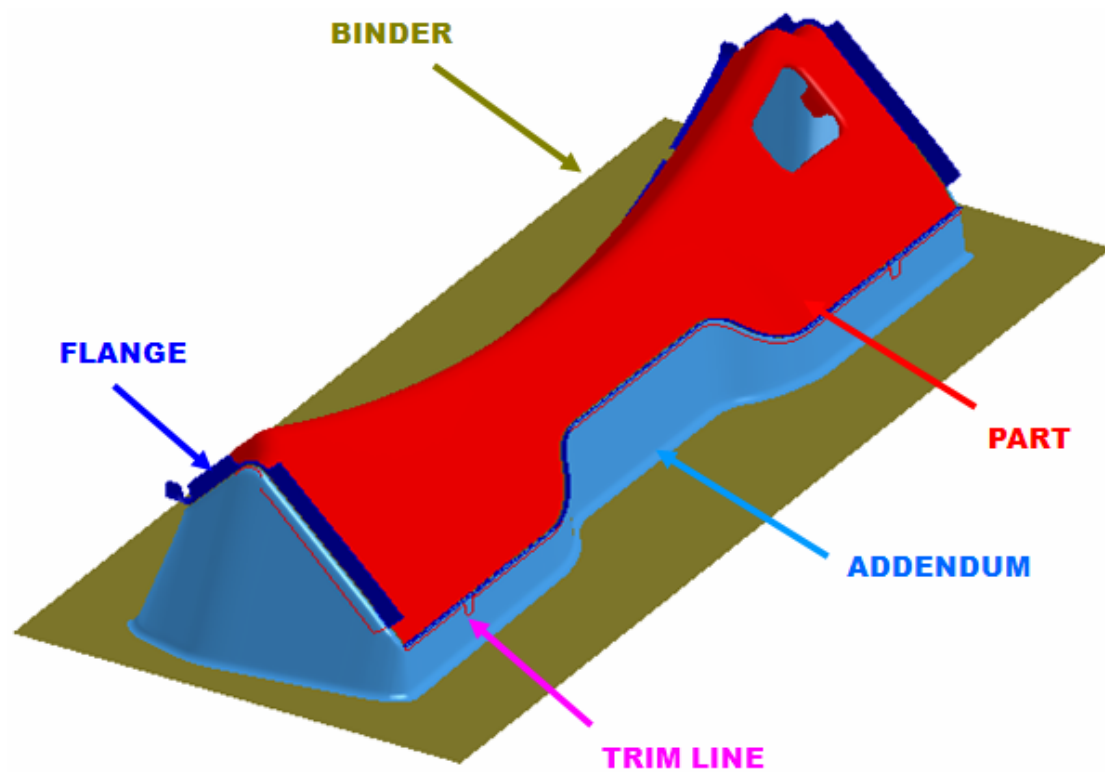


Figure 6.3.43 After Created the Trim Line operation.

#### DELETE TRIM LINE

This function enables the user to remove the trim line.

#### EDIT LINE

All of the functions in this group enable the user to edit the trim line including Insert Points, Move Points and Delete Points.

**Insert Points:** This function enables the user to add points on the trim line. After the function is selected, the user selects a line. All of the points on the line are highlighted. The user can click the location on the screen with the left arrow. The point will be added on the line according to the shortest length to the pick position.

**Move Points:** This function enables the user to move the points on the trim line to a new location. After the function is selected, the user selects a line. All of the points on the line are highlighted. The user can select one point on the line following by picking a location on the screen. The selected points are moved to the new position.

**Delete Points:** This function enables the user to remove points on the trim line. After the



function is selected, the user selects a line. All of the points on the line are highlighted. The points are removed.

#### **UPDATE**

This function enables the user to update the trim line if the addendum or the binder is modified. The trim line is updated according to the new addendum.

#### **UNDO**

This function enables the user to UNDO the result from previous operation.

#### **CLOSE**

This function enables the user to exit the TRIM LINE dialog box.

### **6.3.8 CREATE ADDENDUM SURFACE**

This function creates the surfaces based on the profiles and the POP LINE. The created surfaces are stored in the part named ADDENDUM.

## 6.4 RE-ENGINEERING

The Re-Engineering function is a brand new module in eta/DYNAFORM 5.5. One major objective of this function is to enable the user to translate the addendum and binder which are designed by other CAD software to the addendum in eta/DYNAFORM's DFE environment. The user can easily modify the addendum with the tools provided in eta/DYNAFORM environment. Another objective of the Re-Engineering is to translate the addendum and binder design from a part to another part which share similar shape. As a result, the user can save time in die face design.

The dialog box of RE-Engineering is illustrated in Figure 6.4.1.

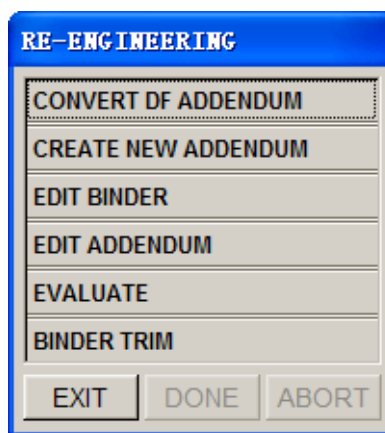


Figure 6.4.1 Re-Engineering dialog box

### 6.4.1 CONVERT DF ADDENDUM

This function allows user to translate the addendum which designed in other CAD software to the addendum that eta/DYNAFORM can be identified. The user can directly modify the addendum by using the tools in eta/DYNAFORM environment. The dialog box of Convert DF Addendum is illustrated in Figure 6.4.2.

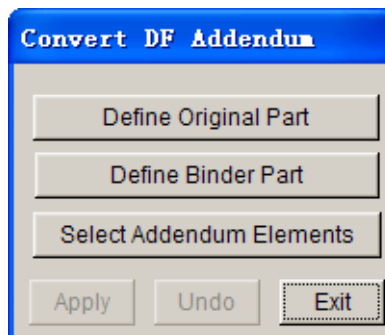


Figure 6.4.2 Convert DF Addendum dialog box

**DEFINE ORIGINAL PART**

The DIE and BINDER must be defined prior to creating addendum. This function enables the user to assign the original part(s) as DIE using the SELECT PART function. If the user has defined the DIE using DEFINE TOOLS function in TOOLS menu, this button is disabled. As illustrated in Figure 6.4.3, the user selects the red part as the original part.

**DEFINE BINDER PART**

This function enables the user to assign BINDER using the SELECT PART function. As illustrated in Figure 6.4.3, the user selects the green part as the BINDER.

**SELECT ADDENDUM ELEMENTS**

This function enables the user to select all the elements of original addendum design. The function provided in SELECT ELEMENTS dialog box is utilized to conduct the operation. As illustrated in Figure 6.4.3, the user selects all elements in the blue part as the addendum part.

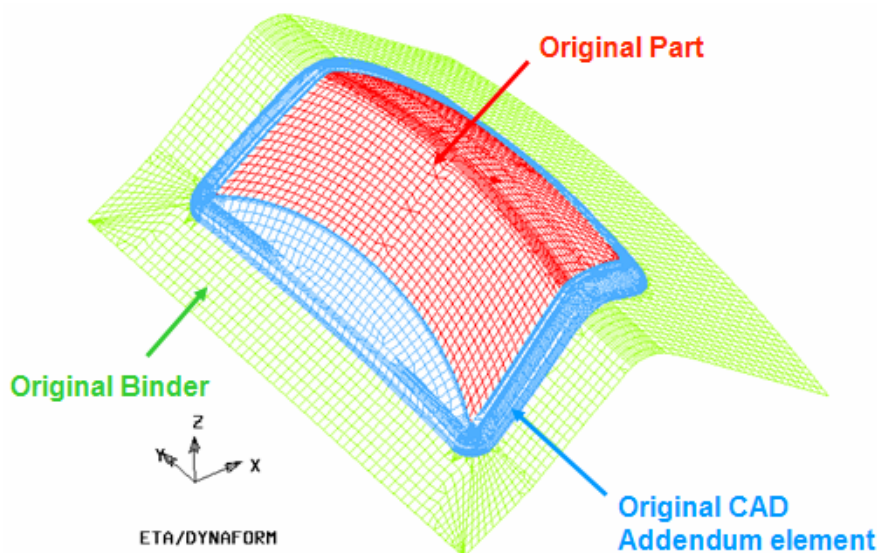


Figure6.4.3 Classification of parts in Re-Engineering

**APPLY**

This function is used to convert the original addendum to the addendum in DFE environment. eta/DYNAFORM creates the new addendum in the original position. A typical result is illustrated in Figure 6.4.4.

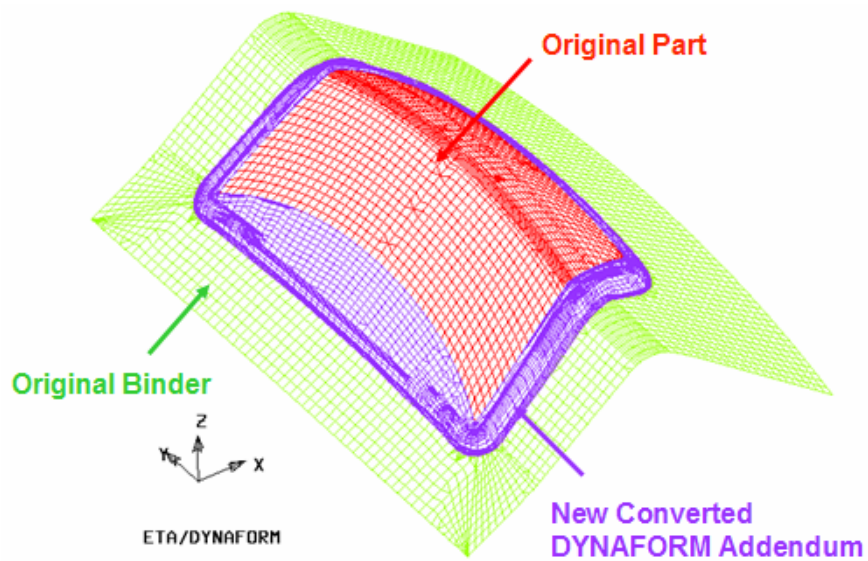


Figure 6.4.4 The new converted addendum in DYNAFORM

#### **UNDO**

Undo the last convert operation.

#### **EXIT**

Dismiss the Convert dialog box

### **6.4.2 CREATE NEW ADDENDUM**

This function enables user to transform the addendum and binder for original part design to another part which shares similar shape. The dialog box of Create New Addendum is illustrated in Figure 6.4.5.

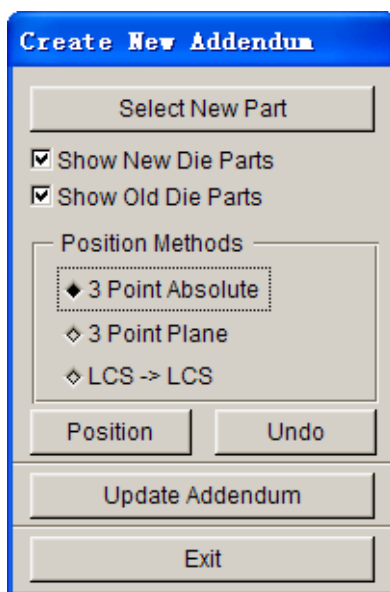


Figure 6.4.5 Create New Addendum dialog box

#### **SELECT NEW PART**

This function enables user to select the new part. The new part will replace the original part as DIE.

#### **SHOW NEW DIE PARTS**

This option is utilized to show/hide the new die.

#### **SHOW OLD DIE PARTS**

This option is utilized to show/hide the original die.

#### **POSITION**

The positioning between the new and original parts is very important to ensure good result. Due to similarity between the new and original parts, the user can use three different methods to position the new part:

- **3 Point Absolute**

Three reference points are selected on the new die parts and original die parts. The first selected reference point on new die is used to position the new die parts. The other two reference points are used to orient the new die part. Three arrows are plotted on screen after the reference points are selected. An illustration is shown in Figure 6.4.6.

**NOTE:** The distance between the selected three points should large enough to ensure good positioning result.

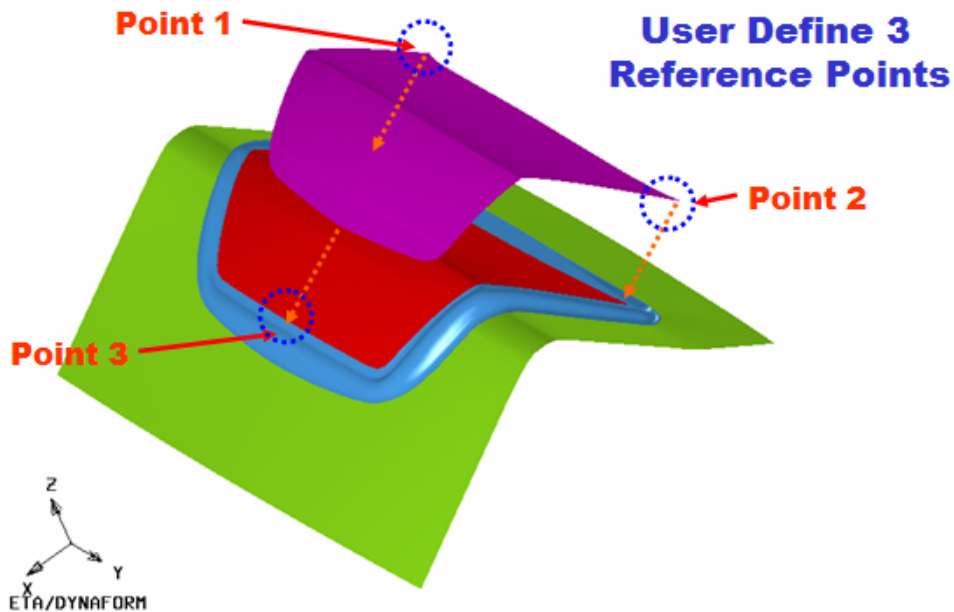


Figure 6.4.6 Three Points Absolute

- **3 Point Plane**

The average of the three selected points (the center point of three points) is used new die parts. The center point of selected point on new die parts is moved to the center point of selected three points on old die part accurately after positioning. The other two points are used to orient the new die part. Three arrows are plotted on screen after the points are selected.

- **LCS→LCS**

This method is used to position the new die parts through defining LCS on new and original die parts. The LCS on new die parts is matched with the LSC on original die parts to position the new and original die parts.

### **UNDO**

Undo the last position operation.

### **UPDATE ADDENDUM**

This function enables user update the addendum of new die parts. An illustration of updated addendum is shown in Figure 6.4.8.

Due to minor differences between the new and original die part, some gaps or overlap elements may exist between the original addendum and the new die part. The user may need to adjust some points on original addendum to the new die parts. After the function is selected, some arrows are displayed from point on original addendum to the new die part. The user has to adjust the corresponding point by selecting the head of an arrow. Then, the user can select a new position on screen to change the corresponding point. Figure 6.4.7 illustrates the position of corresponding points.

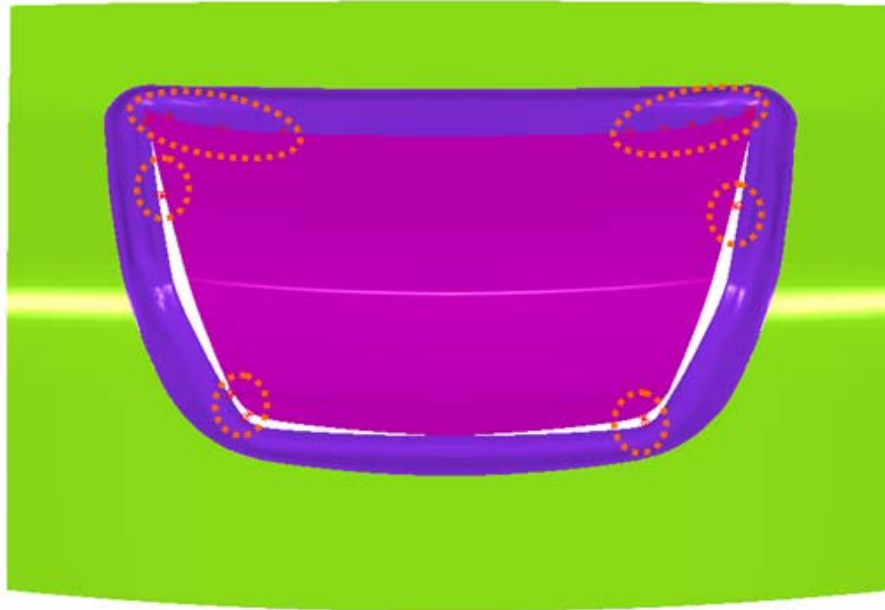


Figure 6.4.7 Moving the corresponding points

After the adjustment is completed, the user can click on middle mouse button to update the addendum on new die parts.

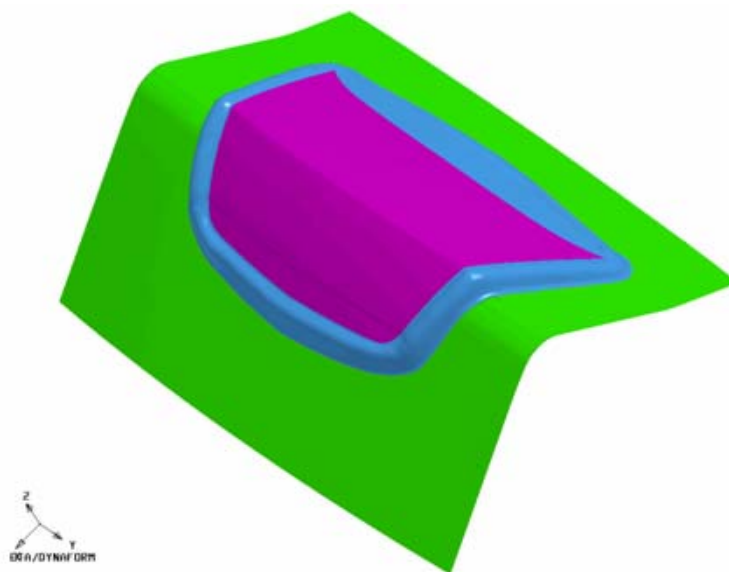


Figure 6.4.8 Updated addendum

### 6.4.3 EDIT BINDER

This function enables user to edit the original trimmed binder. The inner hole of the trimmed binder will be filled with elements. The result of Edit Binder is illustrated in Figure 6.4.9.

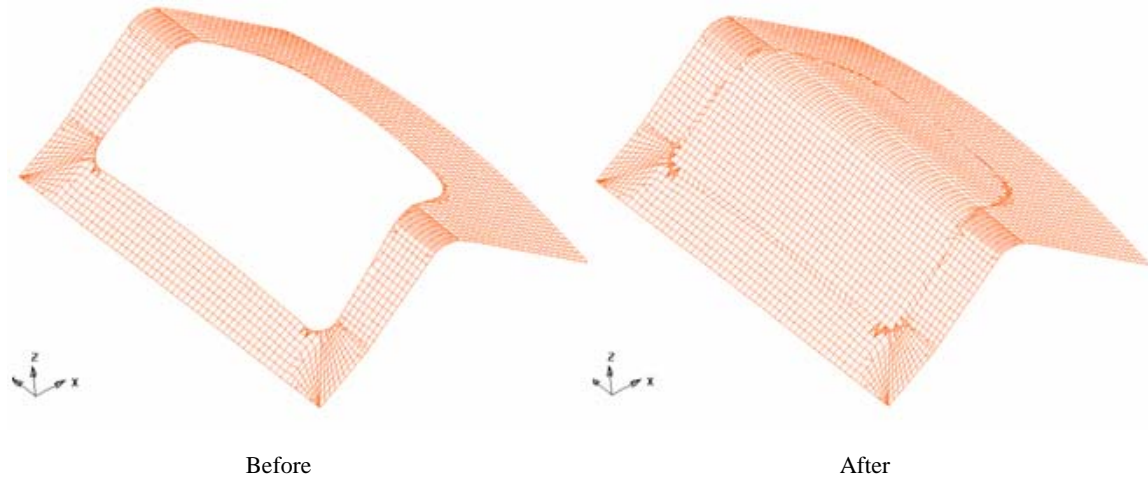


Figure 6.4.9 Edit Binder

### 6.4.4 EDIT ADDENDUM

This function provides a shortcut for user to access the ADDENDUM dialog box without exiting the Re-Engineering dialog box. User can do further modification for the new addendum. Please refer to the section 6.3.3 for information about editing addendum.

### 6.4.5 EVALUATE

This function enables user to evaluate the error between elements of the original addendum and those of the new addendum. The error is displayed on the display area with color contour. The EVALUATE dialog box is illustrated in Figure 6.4.10.

#### FIRST ELM SET

Enable user to select the first set of elements for evaluation.

#### FIRST ELM SET

Enable user to select the second set of elements for evaluation.



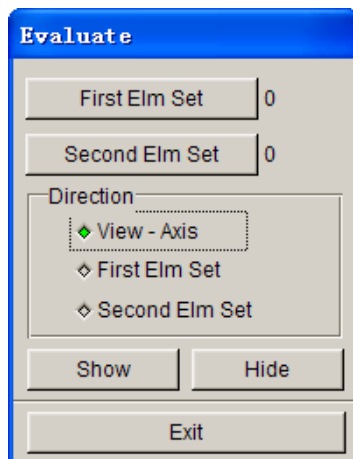


Figure 6.4.10 Evaluate dialog box

### DIRECTION

This function enables user to define the direction which is used to evaluate the error between two sets of elements.

- **View-Axis:** Set the View direction as the direction to evaluate the error between two sets of elements.
- **First Elm Set:** Set the normal direction of first set of elements as the direction to evaluate the error between two sets of elements.
- **Second Elm Set:** Set the normal direction of the second set of elements as the direction to evaluate the error between two sets of elements.

### SHOW

After two set of elements are selected and the direction is defined, the user can click this button to show the result. Figure 6.4.11 illustrates the result after Evaluate operation.

### HIDE

This button enables user to hide the contour of the error.

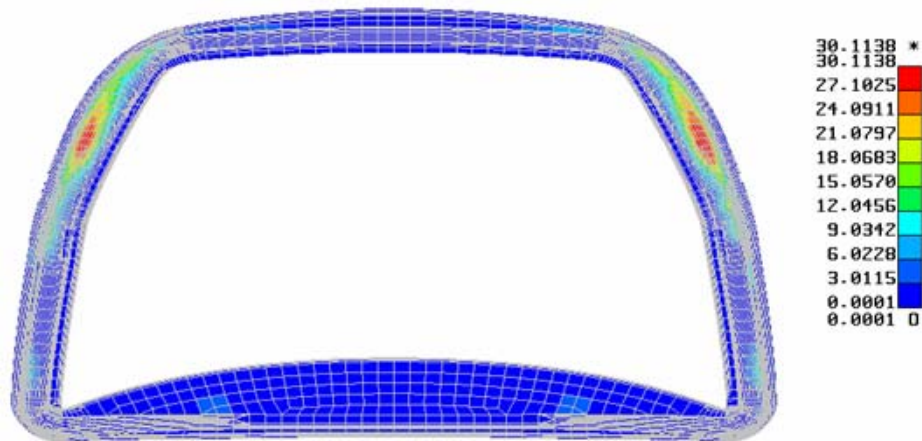


Figure 6.4.11 Contour of the Evaluate

#### 6.4.6 BINDER TRIM

After creating the new addendum, the user has to trim the binder. This function provides the user a shortcut to access the Binder Trim function. Please refer to section 6.5.7 for information about the Binder Trim function.

### 6.5 MODIFICATION

The functions provided in this menu are used to modify lines, surfaces or elements and to complete the die face design. The DFE MODIFICATION dialog box is illustrated in Figure 6.5.1.

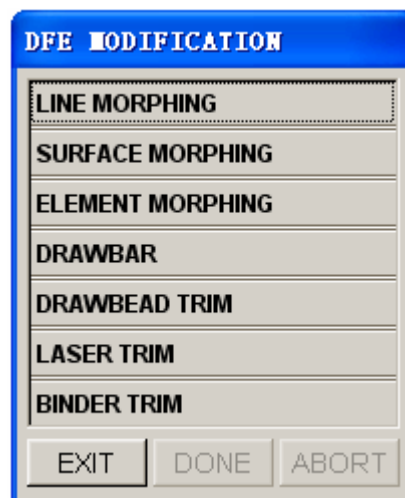


Figure 6.5.1 DFE Modification dialog box

### 6.5.1 LINE MORPHING

This function is used to modify a line by moving a control point on the line. The morphing operation is described in the following steps.

1. Select a line using the function provided in the SELECT LINE dialog box.
2. After a line is selected, the LINE MORPHING dialog box as illustrated in Figure 6.5.2 is displayed.

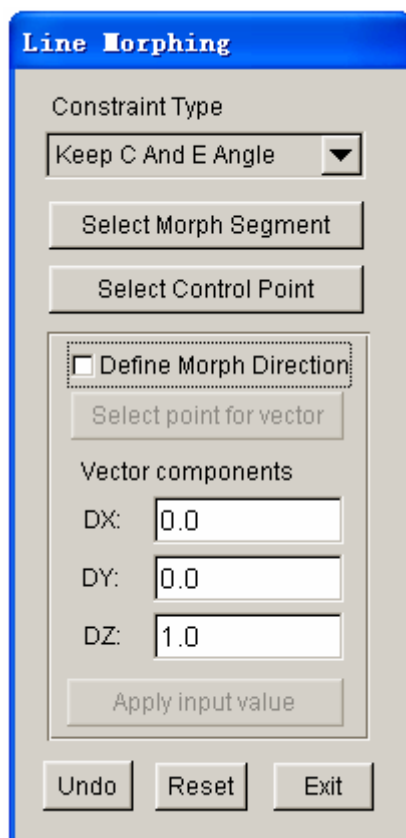


Figure 6.5.2 Line Morphing dialog box

- **CONSTRAINT TYPES**

There are three types of constraint in line morphing: NO CONSTRAINT, KEEP C ANGLE and KEEP E ANGLE. For a detailed description of the morphing types, refer to Section 6.2.1. Click on the drop down button to select the type of constraint.

- **SELECT MORPH SEGMENT**

This option allows the user to define a segment of the selected line to morph by selecting two points on the line. If no segment is defined, the entire selected line is

morphed.

- **SELECT CONTROL POINT**

The user must select a control point on the line. Click on the left mouse button to select new control point, middle mouse button to use the previous control point, or right mouse button to cancel the selection. Once the control point is selected, the default morph vector is displayed at the control point. Move the mouse to morph the line. The user can view the morphed result in dynamic mode. When the desired result is obtained, click on the left mouse button again to complete the morphing operation or click on the right mouse button to cancel the operation. If the user wants to define another morphing direction, toggle on checkbox of the DEFINE MORPH DIRECTION option.

- **DEFINE MORPH DIRECTION**

This option allows user to define a new morphing direction. Once this option is selected, the SELECT POINT FOR VECTOR button is enabled. The user define the direction using function provided in the INPUT COORDINATE dialog box. Click the SELECT POINT FOR VECTOR button to select a point or node on the display area. The new morphing direction is the vector from the control point to the selected point. The user may enter the vector components directly in the VECTOR COMPONENTS dialog box, following by clicking on the APPLY INPUT VALUE button to define the vector.

UNDO allows the user to reject the result from the last morphing operation.

RESET returns the selected line to the original shape after a series of morphing operations.

3. Click on EXIT button to complete the morphing operation. The SELECT LINE dialog box is displayed to enable the user to select another line for morphing.
4. Click on CANCEL button in the SELECT LINE dialog box to exit the function.

## 6.5.2 SURFACE MORPHING

This menu allows the user to morph the selected surface. The dialog box is illustrated in Figure 6.5.3.



Figure 6.5.3 Surface Morphing dialog box

### 1. INTERIOR MORPHING

This function allows the user to morph a surface by selecting a control point within the surface domain. The user may morph the entire surface or a region on the surface.

- Select a surface using the function provided in SELECT SURFACE dialog box.
- After a surface is selected, the dialog box as illustrated in Figure 6.5.4 is displayed.

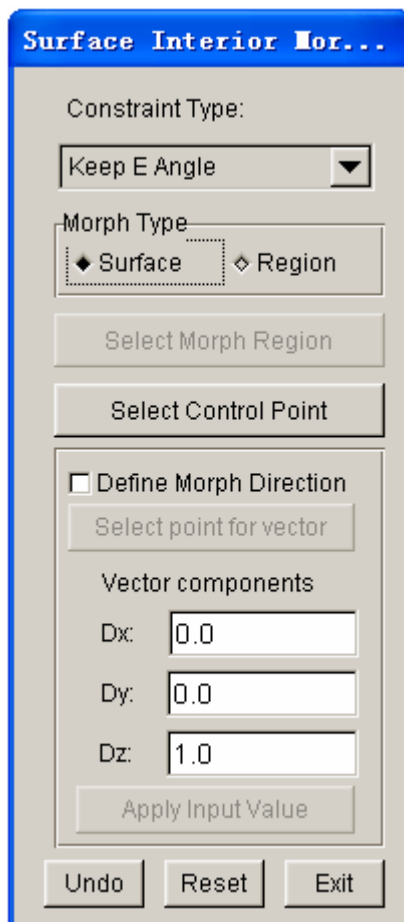


Figure 6.5.4 Surface Morphing – Interior

- The drop down button allows the user to select the type of constraint for morphing. The description of each constraint types is discussed in Section 6.2.1.
- If REGION is selected, the SELECT MORPH REGION button is enabled for the user to select two diagonal points on the surface to define the morphing region.
- Click on the SELECT CONTROL POINT button to select a point on the surface (or in the region if the REGION option is selected) as the control point. After point is selected, a morphing vector and U-V line are plotted at the control point. The user may click on the left mouse button following by moving the cursor to view the morphed U-V line in dynamic mode. When a desired shape is obtained, the user clicks on the left mouse button to accept the morphed surface. The surface is updated according to the new U-V lines.
- The user may toggle on checkbox of the DEFINE MORPH DIRECTION option to define a new morphing direction. Detailed description about this option is discussed in Section 6.5.1.
- UNDO aborts the last morphing operation. RESET sets the surface back to its original shape.

## 2. EDGE MORPHING

This function uses a point on the surface boundary as the control point. It operates similarly to the INTERIOR MORPHING function.

## 3. CORNER MORPHING

This function uses a corner point on the surface as the control point. It operates similarly to the INTERIOR MORPHING function.

## 4. SECTION LINE MORPHING

This function enables the user to modify surfaces by morphing a U or V section line on the surface.

- Select a surface by using the function provided in SELECT SURFACE dialog box.
- After a surface is selected, the U-V lines are displayed on the surface and a dialog box as illustrated in Figure 6.5.5 is displayed.

- Select the CONSTRAINT TYPE as described in the INTERIOR MORPHING function.
- Click on the SELECT SECTION LINE button to select a U or V section line as control line.

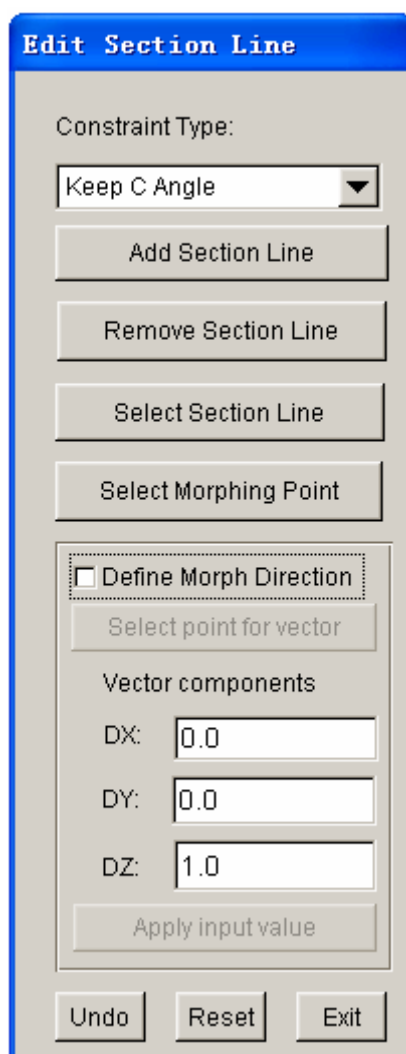


Figure 6.5.5 Section Line Morphing dialog box

- Click on the SELECT MORPHING POINT button to select a control point on the selected section line. After an interior point is selected, the user may move the cursor to view the morphed U-V line in dynamic mode. When a desired shape is obtained, the user clicks on the left mouse button to accept the morphed result. The surface is updated according to the new control line.
- Toggle on checkbox of the DEFINE MORPH DIRECTION option to define a morphing vector.

- ADD SECTION LINE function allows the user to add UV section lines at the cursor location. REMOVE SECTION LINE allows the user to remove U or V section lines using mouse pick. UNDO aborts the last morphing operation. RESET sets the surface back to its original shape.

### 6.5.3 ELEMENT MORPHING

This menu allows the user to morph a region of selected elements. As illustrated in Figure 6.5.6, four types of morphing approaches are provided. All elements within the region must be selected without excluding any elements from the region. The operation of these functions is similar to those described in the SURFACE MORPHING.



Figure 6.5.6 Element Morphing dialog box

#### 1. INTERIOR MORPHING

This function allows the user to morph the selected elements by using a node within the element group domain as the control point.

- After a group of elements is selected, the following message is printed in the message prompt window:

***SELECT CONTROL NODE FOR MORPHING***

Select any interior node from the selected element group.

- After the node is selected, the dialog box as illustrated in Figure 6.5.7 is displayed.



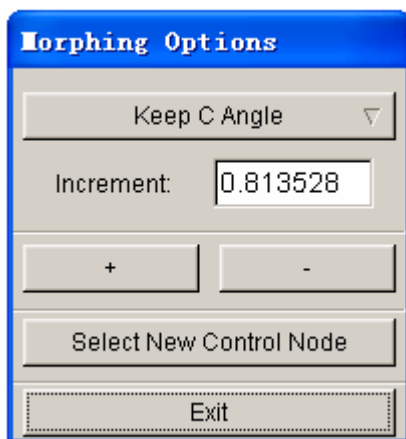


Figure 6.5.7 Morphing dialog box

The drop down button allows the user to select type of constraint for morphing. The description of each constraint type is discussed in Section 6.2.1.

After a constraint type is selected, enter the morph increment in the dialog box. Click on the + or - button to morph the surface. The user may use the NEW CONTROL NODE option to select another interior node on the surface as the control node. Figure 6.5.8 illustrates a typical result (red) generated by element interior morphing.

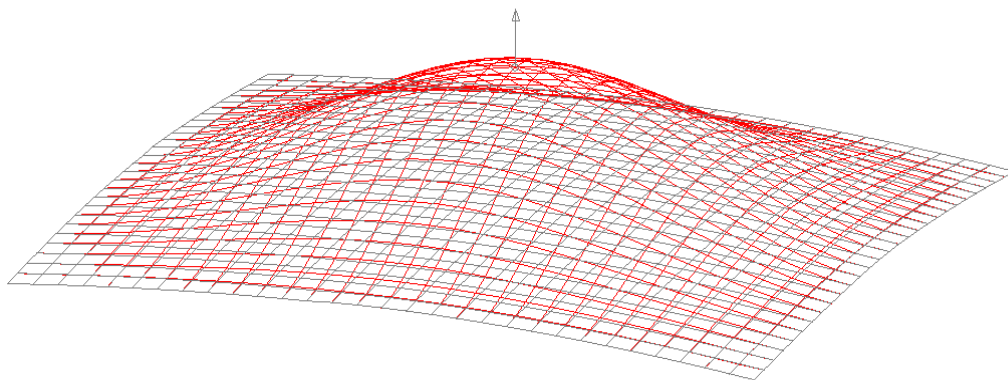


Figure 6.5.8 An illustration showing a typical result generated by Element Interior Morphing

- Click on EXIT button to update the morphed elements and display the SELECT ELEMENTS dialog box.
- Select other element to repeat the operation or click on CANCEL button to exit the function.

## 2. EDGE MORPHING

This function allows the user to select a node on the boundary of the element region as the control node. The operation is similar to those described in the INTERIOR MORPHING.

## 3. CORNER MORPHING

This function allows the user to select a corner node of the element region as the control node. The operation is similar to those described in the INTERIOR MORPHING.

## 4. EDGE QUADRATIC

This function enables the user to morph a region of elements controlled by an edge of the region. The selected elements must include distinctive corner features along the morphing edge. The user must select a node along the morphing edge as the control node. The entire edge is selected as the control line to morph the elements in the region.

The rest of the operations are similar to those described in INTERIOR MORPHING.

### 6.5.4 DRAWBAR

The drawbar menu provides functions to generate drawbars on the die surface. There are three types of drawbars available:

1. DRAW BAR ENDS WITH A SPHERICAL SHAPE (TEAR DROP HEAD). Figure 6.5.9 shows the control parameters in the dialog box.
2. DRAW BAR ENDS WITH A CONICAL SHAPE (TEAR DROP TAIL). Figure 6.5.10 shows the parameters in the dialog box.

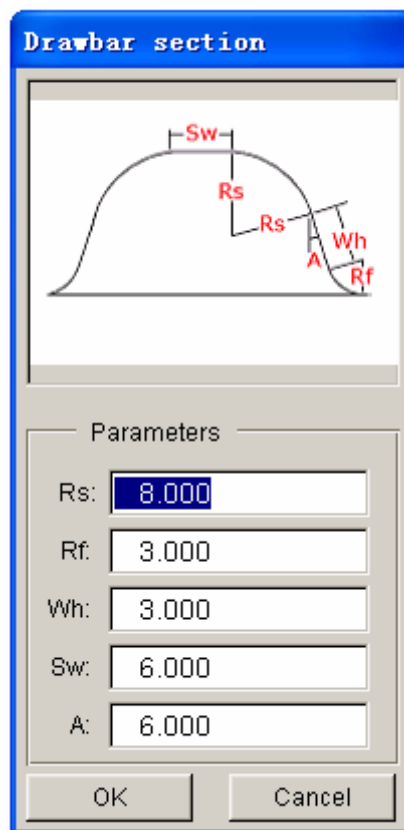


Figure 6.5.9 Parameters for Tear Drop Head

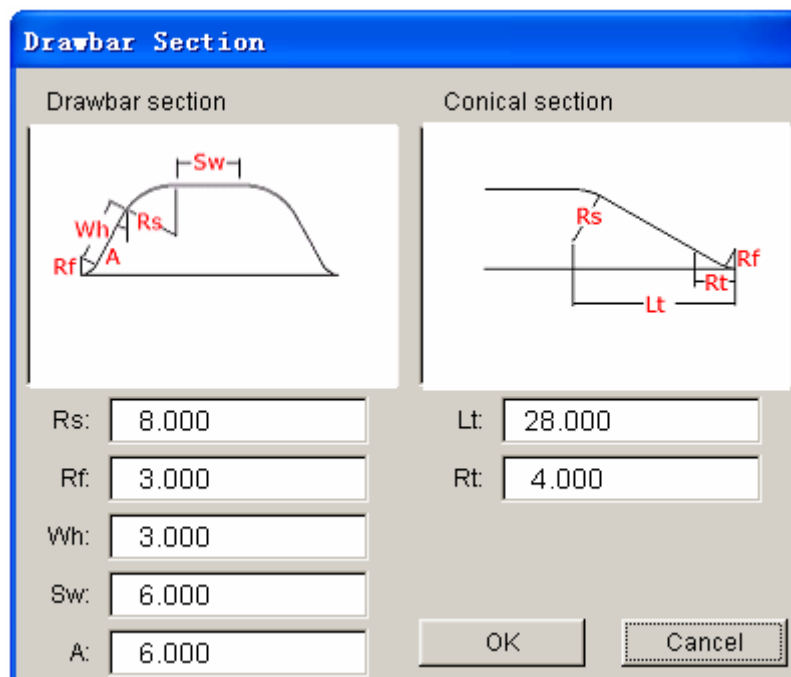


Figure 6.5.10 Parameters for Tear Drop Tail

3. TEAR DROP (DRAW BAR WITHOUT MAIN BODY). Figure 6.4.11 shows the parameters in the dialog box.

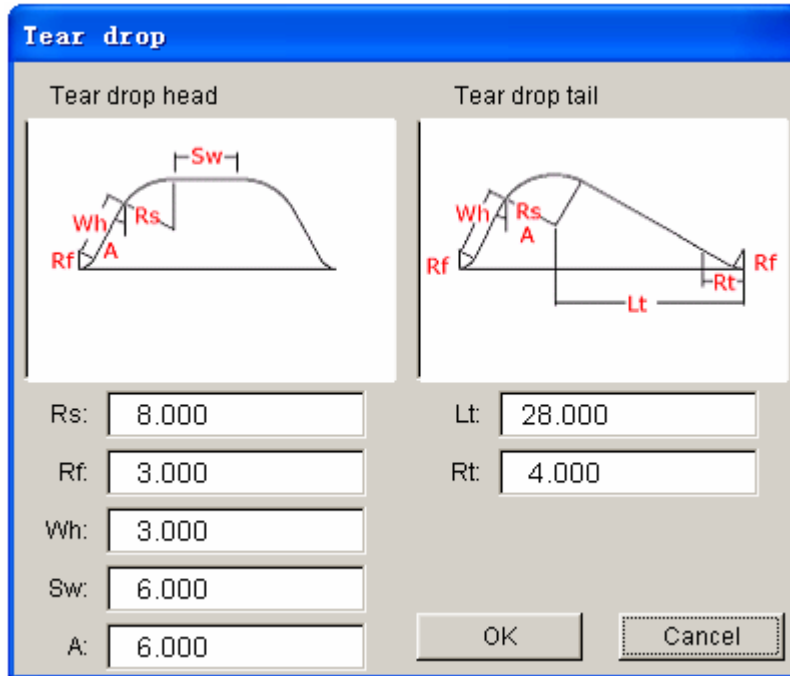


Figure 6.5.11 Parameters for Tear Drop

Control Parameters:

- Rs:** Drawbar Radius
- Rf:** Fillet Radius
- Wh:** Wall Height (can be zero)
- Sw:** Top Strip Width (can be zero)
- A:** Wall Angle (default of 6 degrees)
- Rt:** Tail Circle Radius
- Lt:** Length of Tear Drop

The steps to create a drawbar are listed as following:

- Click on the DRAWBAR function from the DFE MODIFICATION dialog box to display the DRAWBAR dialog box as illustrated in Figure 6.5.12a. If the user wishes to define the element size of the drawbar, toggle off checkbox of the AUTO ELEMENT SIZE option. As illustrated in Figure 6.5.12b, the user can enter the element size in the input field.
- Select one type of drawbar. The default selection is DRAW BAR ENDS WITH A SPHERICAL SHAPE (TEAR DROP HEAD).

- Click on the PARAMETERS button to display the corresponding dialog box as illustrated in Figures 6.5.9-6.5.11.

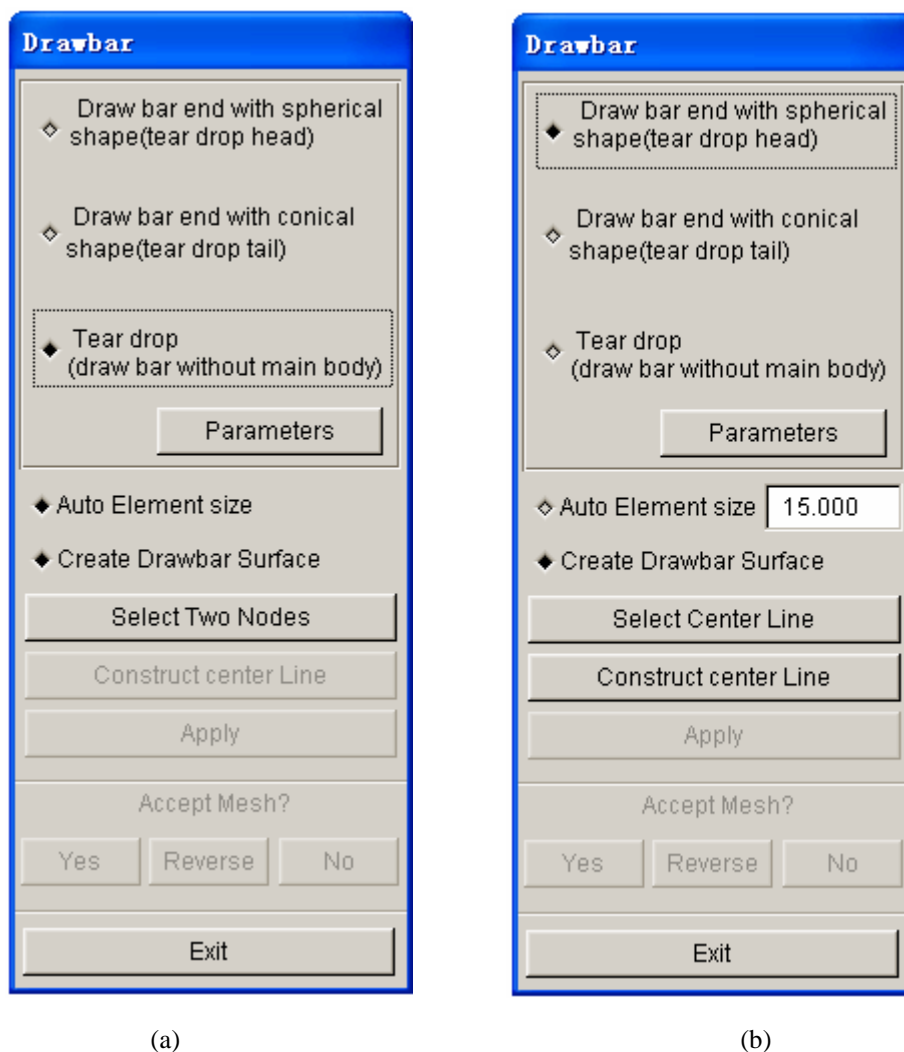


Figure 6.5.12 Drawbar dialog box (a) With Auto Element Size (b) With User Defined Element Size

- The CREATE DRAWBAR SURFACE option enables the user to generate drawbar surfaces. This option is selected as default. The user may disable the option if the drawbar surfaces are not required.
- Define the centerline of the drawbar. The centerline is the center of the drawbar's main body, including both ends.
- If centerline is available in the current database, click on the SELECT CENTER LINE button to select the centerline. Else, click on the CONSTRUCT CENTER LINE button to create a line by selecting nodes on the mesh.
- After the drawbar parameters and centerline are defined, click on the APPLY button to

generate the drawbar. Select YES to keep the drawbar mesh or NO to reject the mesh. The REVERSE button is used to reverse the drawbar on the opposite side of the tool. If the CREATE DRAWBAR SURFACE option is selected, the drawbar surface will be generated together with the mesh.

If the tear drop drawbar is selected, the dialog box as illustrated in Figure 6.5.13 is displayed. Click on the SELECT TWO NODES button to select two nodes to define the direction of the centerline. The head of the tear drop will be placed at the first selected node. The length of the tear drop is determined by the parameters defined in the PARAMETER window.

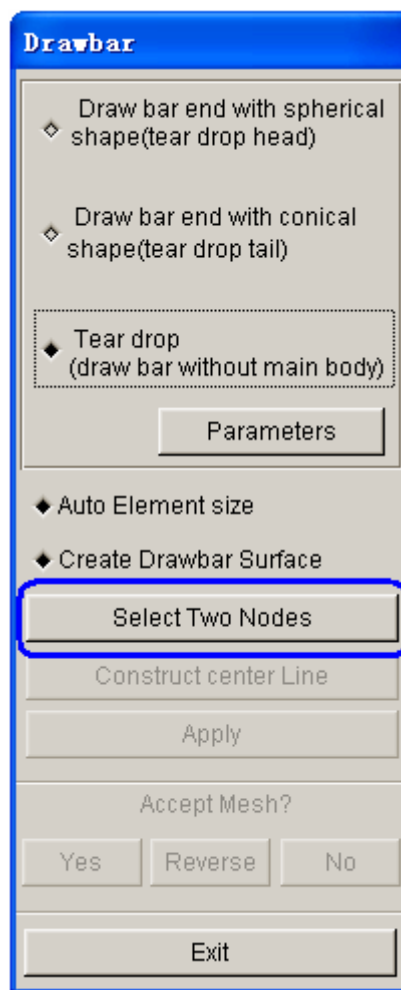


Figure 6.5.13 Tear Drop

Figure 6.5.14 shows the details of three types of drawbars.

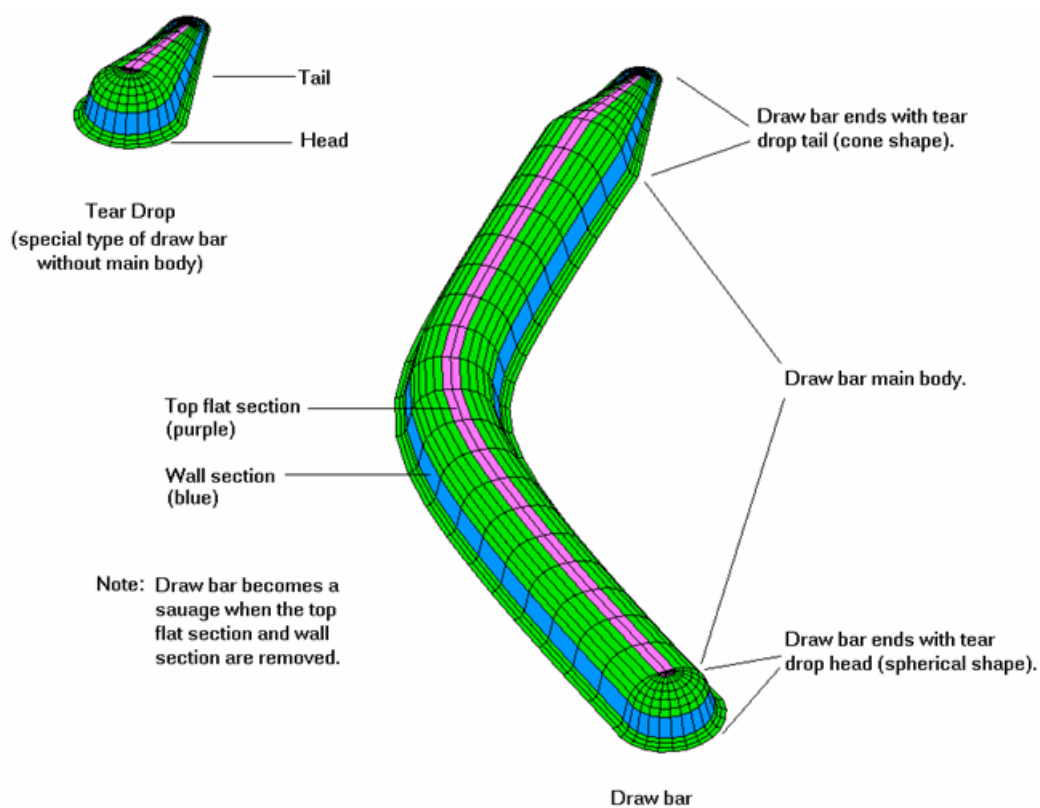


Figure 6.5.14 Three Types of Drawbars

### 6.5.5 DRAWBEAD TRIM

The DRAWBEAD TRIM function is used to trim the mesh along the outline of the drawbead. The operation of this function is described as following:

- The user select drawbead outline using the function provided in SELECT LINE dialog box. The drawbead outline is defined by one or more than one lines.
- Select a line as illustrated in Figure 6.5.15.
- Select OK to accept the selected line. The trimmed result is displayed along with a DYNAFORM QUESTION box as illustrated in Figure 6.5.17.
- Click YES to accept the mesh.

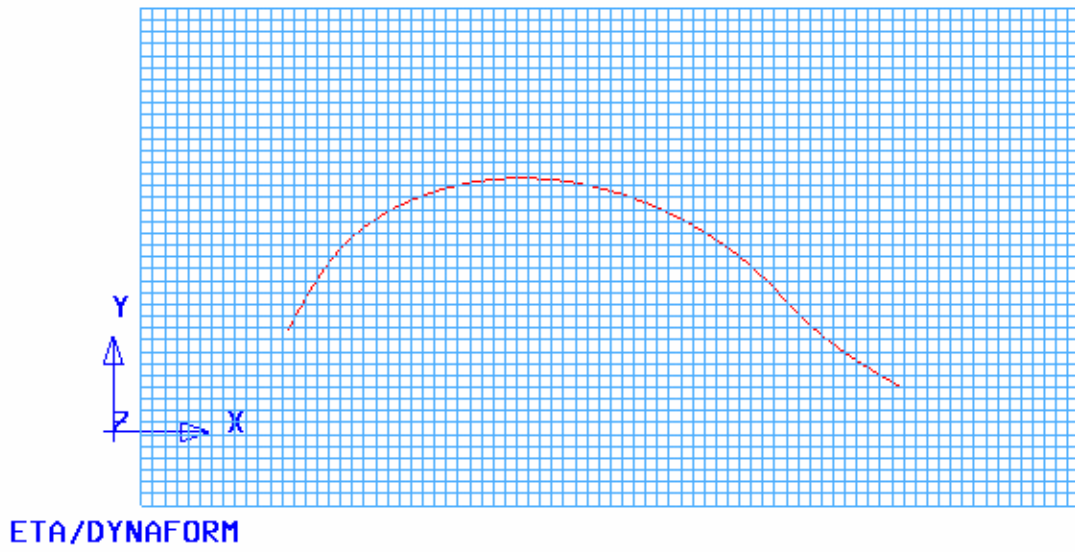


Figure 6.5.15 Select Drawbead Outline

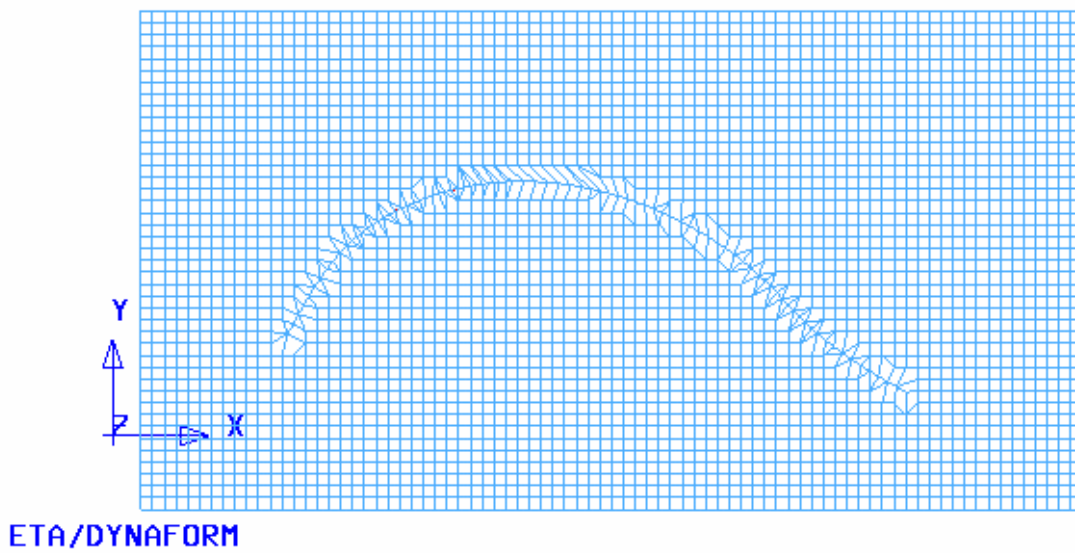


Figure 6.5.16 Trim Result

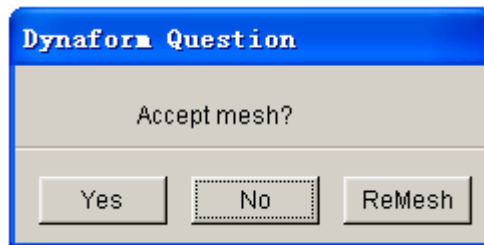


Figure 6.5.17 DYNAFORM QUESTION dialog box



### 6.5.6 LASER TRIM

This function is used to trim the part from a line normal to the part. The selected line is considered as the cutting path of laser beam. The operation of this function is similar to those described in DRAWBEAD TRIM with an exception that a closed curve is used as the trim line.

### 6.5.7 BINDER TRIM

After the addendum is generated, the binder is trimmed and connected to the die. This function allows the user to trim the binder to complete the die face design. If the addendum is defined for the die, the POP LINE is used to trim the binder. If addendum is not defined, the die boundary line will be used to trim the binder. The BINDER TRIM dialog box is illustrated in Figure 6.5.18.

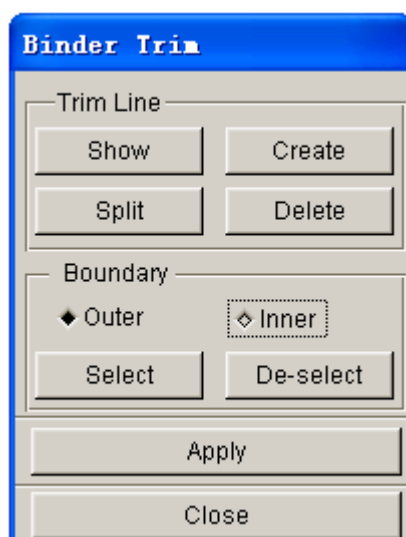


Figure 6.5.18 Binder Trim dialog box

- **EDIT LINE**

These functions enable the user to edit the trim line. Refer to Section 6.2.1 for a detailed description of EDIT LINE functions.

- **BOUNDARY**

There are two types of boundary available for trimming operation. The trim line can be used as an inner boundary or outer boundary. If OUTER is toggled on, the part outside of the trim line is kept after trimming. If INNER is toggled on, the part within the trim line is kept after trimming. If there is an inner addendum, for example, the inner binder trim is required.

- **SELECT and DESELECT**

The SELECT button enables the user to select a line from the current database as the trim line.

The DESELECT button enables the user to deselect the selected line.

- After the trim line is selected, click on the APPLY button to trim the binder.

## 6.6 DIE DESIGN CHECK

This function allows the user to visually check the die design according to the minimum tip angle and minimum draw depth. The dialog box is illustrated in Figure 6.6.1.

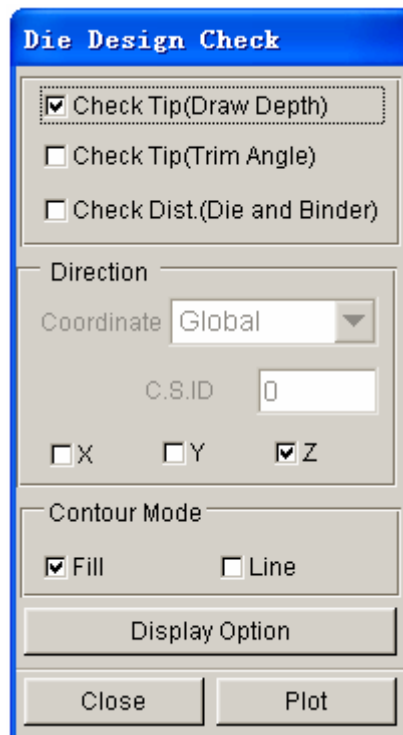


Figure 6.6.1 Die Design Check dialog box

### CHECK TIP (DRAW DEPTH)

This option is based on a flat blank. The function is similar to the draw depth contour display of tipping function described in Section 6.1.9.

Click on the PLOT button to display the draw depth contour for the die design.

**CHECK TIP (TRIM ANGLE)**

This option calculates and displays the trim angle of a part at typically 30 degrees to the forming direction. It checks the maximum angle for direct trim.

Click on the PLOT button to display the contour of the trim angle. Red color indicates the area needs cam trim, while blue indicates direct trim.

**CHECK DISTANCE (DIE AND BINDER)**

This function checks the draw depth relative to the binder surface. The binder must be defined in the database prior to performing CHECK DISTANCE operation. This function does not work if the binder surface has been trimmed by the BINDER TRIM function.

Click on the PLOT button to display a distance contour for the distance between the die and binder.

**DIRECTION**

This function enables the user to select a draw direction; Global Coordinate System (GCS) or Local Coordinate System (LCS).

**DISPLAY OPTION**

This function controls the result display. Refer to the eta/POST USER'S MANUAL for a detailed description of the display options.

## Chapter 7

### BSE (BLANK SIZE ENGINEERING)

BLANK SIZE ENGINEERING (BSE) is an eta/DYNAFORM add-on module. It includes an advanced one step code MSTEP for fast stamping simulation. In addition, BSE can be utilized to estimate a blank size and conduct a blank development, such as blank fitting and blank nesting. BSE submenu consists of PREPARATION, MSTEP, and DEVELOPMENT, as shown in Figure 7.1.



Figure 7.1: BSE menu

A detailed description of each submenu and its corresponding functions is given in the following sections.

## 7.1 PREPARATION

As shown in Figure 7.1.1, there are several functions in the PREPARATION dialog box. The user may start with import part geometry or part mesh using BSE/PREPARATION/IMPORT OR FILE/IMPORT. Refer to Section 3.6 for details on importing geometry files.

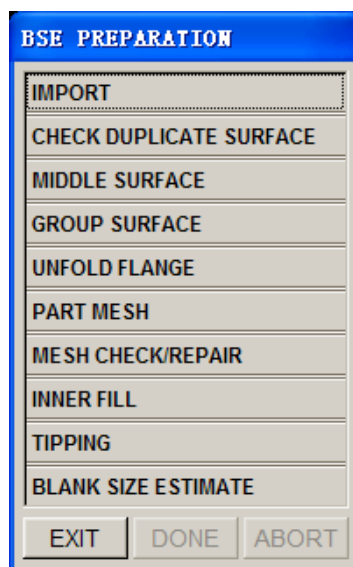


Figure 7.1.1 BSE Preparation dialog box

The CHECK DUPLICATE SURFACE function enables the user to check and highlight duplicate surfaces and include duplicate surfaces in a new part.

The MIDDLE SURFACE function allows user to create the middle surfaces from a solid part. For detailed information, refers to Section 5.2.23.

The GROUP SURFACE function allows the user to separate the TOP and BOTTOM surfaces from a solid part. Detailed information about this function is described in Section 5.2.24.

The UNFOLD FLANGE function enables the user to unfold the flanges fully or by incremental angles on a product. Refer to Section 6.1.2 for detailed information about unfold flange operation.

The PART MESH function enables the user to mesh the part geometry using PART MESHER. Detailed information on PART MESHER is provided in Section 5.3.4.

The MESH CHECK/REPAIR function launches the Mesh Check & Repair dialog box. Detailed information about MESH CHECK/REPAIR is provided in Section 5.5.

The INNER FILL function is used to fill in any holes in the part mesh. Refer to Section 6.1.9 for more details.

Refer to Section 6.1.11 for more information about TIPPING.

### 7.1.1 BSE (BLANK SIZE ESTIMATE)

As shown in Figure 7.1.2, the BLANK SIZE ESTIMATE dialog box enables the user to define material properties and thickness prior to the calculation of the blank outline. The estimated blank outline will be stored in a new part named BO\_LINE. An example of BLANK SIZE ESTIMATION is shown in Figure 7.1.3.

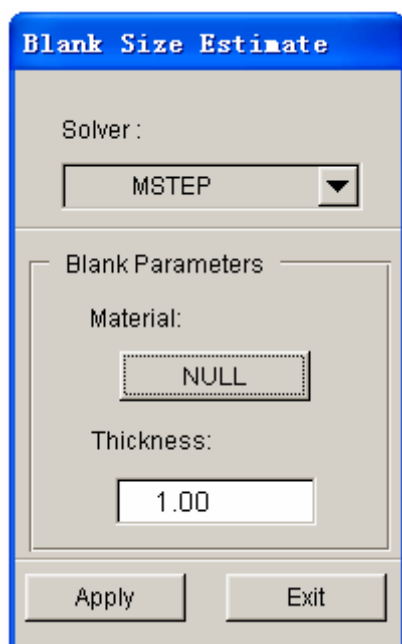


Figure 7.1.2 The Blank Size Estimate dialog box

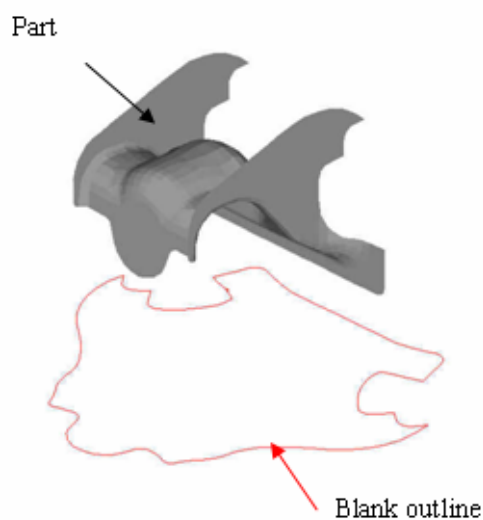


Figure 7.1.3 An example of Blank Size Estimation

Estimation

### SOLVER

There are two solvers available for Blank Size Estimate as shown in Figure 7.1.4:

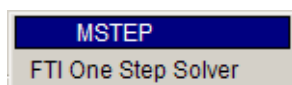


Figure 7.1.4 Solver for Blank Size Estimate

**MSTEP** is a new Modified One Step solver developed by ETA. See detail description in MSTEP section in this chapter.

**FTI's One Step Solver** is a conventional one step solver provided by FTI. It is under the trademark of FTI Corporation, Oakville, Ontario, Canada.

## 7.2 MSTEP

The MSTEP is a new Modified-One-Step solver developed by ETA. It provides an option for iterative solution procedure to achieve more accurate result with trade off of longer computing time. It is mainly used to quickly assess formability and estimate the blank outline in the early stage of automobile body design cycle.

Figure 7.2.1 shows the main interface of MSTEP. The method of defining the tools is similar to method in Quick-Setup menu to define tools. The Sheet part is required input that is the final designed part geometry. These tools marked with blue color are optional input tools.

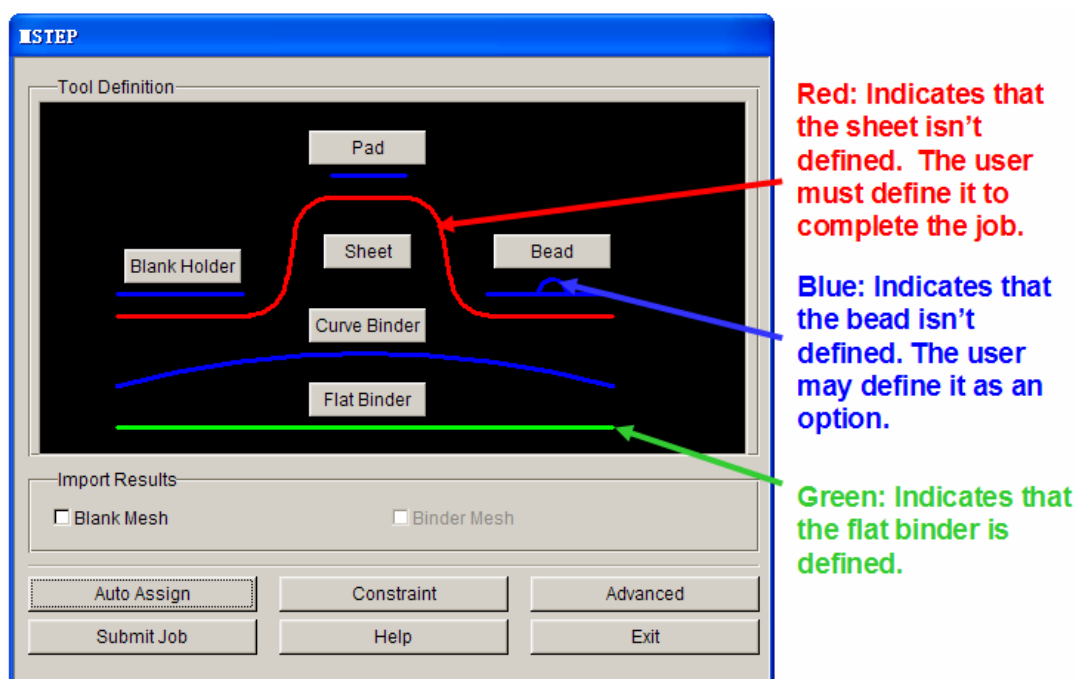


Figure 7.2.1 The Analysis GUI for MSTEP

**AUTO ASSIGN:** The program automatically defines the tools and the sheet.

**CONSTRAIN:** Defines the constraints for the blank. Please refer to Preprocess / Boundary Condition / CONSTRAINTS for Constraint set.

**ADVANCED:** Allows the user to edit some default parameters that relate to MESEP settings.

Figure 7.2.2 shows the interface of MSTEP Setting.

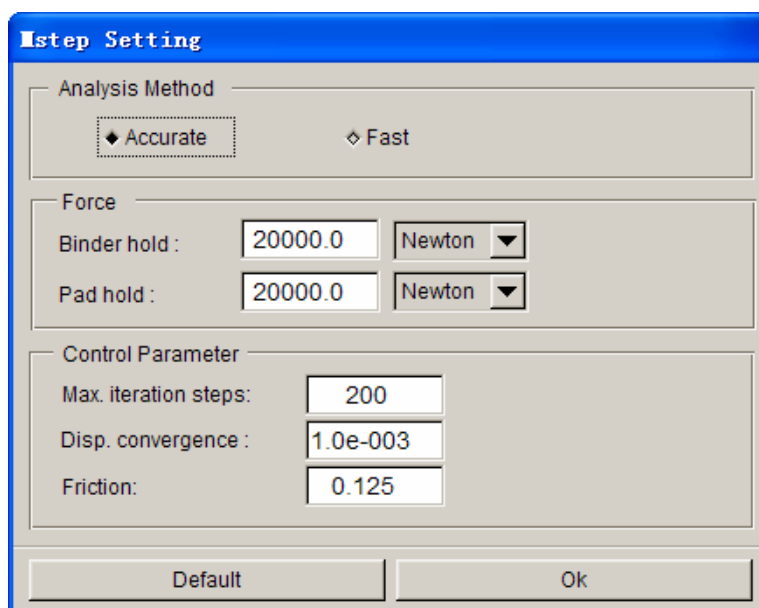


Figure 7.2.2 MSTEP Setting

In the Mstep Setting dialog box, user can select the Analysis Method. There are two different analysis methods in Mstep – Accurate and Fast. In accurate, not only the plasticity and elastic deformation of blank are considered, but also all the practical technique conditions are used. In fast, only the elastic deformation is considered. So the result of accurate method is much better than Fast method. Of course, the solving time of accurate method is larger than the fast method.

### FORCE

**Binder Hold:** Define the Binder Hold Force. The default Binder Hold force is 20000 Newton. User can switch the unit of force from Newton to Ton.

**Pad Hold:** Define the Pad Hold Force. The default Binder Hold force is 20000 Newton.

### CONTROL PARAMETER

**Max. Iteration Steps:** Define the Max. iteration steps during the iteration calculation. If the program can not reach convergence in the max. iteration steps, A error message will print in the \*.out file to notify user. The default max. iteration steps is 200.

**Disp. Convergence:** Define the displacement convergence for the iteration calculation. The default displacement convergence is 0.001.

**Friction:** Define the friction between the blank and tools. The default Friction factor is 0.125. User can change it according to the practical technique conditions.

**HELP:** Provides useful tips for stamping simulation.

**SUBMIT JOB:** Allows the user to submit the current job to the MSTEP solver.



**EXIT:** Allows the user to exit the MSTEP interface with submitting the simulation.

## 7.3 DEVELOPMENT

After obtaining the estimated blank outline, BLANK DEVELOPMENT enables the user to fine-tune the blank outline for subsequent application in blank nesting, cost estimation, and forming simulation. The BLANK DEVELOPMENT dialog box provides several functions for convenient manipulation of the blank outline. The dialog box is shown in Figure 7.3.1.



Figure 7.3.1 The Blank Development dialog box

### 7.3.1 BLANK GENERATION

This function enables the user to generate a flat blank mesh using the blank outline. The elements generated using BLANK GENERATION is QUAD (quadrilateral) dominant (as shown in Figure 7.3.2) which is required for an incremental stamping simulation using LS-DYNA.

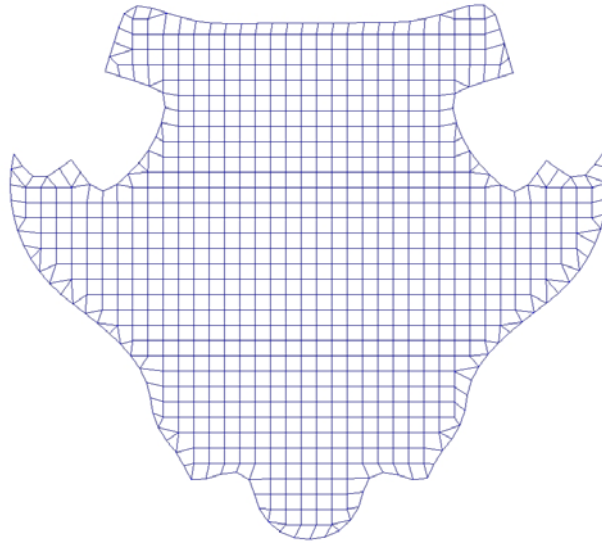


Figure 7.3.2 Blank mesh generated by Blank Generator

### 7.3.2 OUTER SMOOTH

The OUTER SMOOTH functions enable the user to smooth the blank boundary. Refer to Section 6.1.12 for more information about the OUTER SMOOTH functions. An example of smoothing blank boundary is shown in Figure 7.3.3.

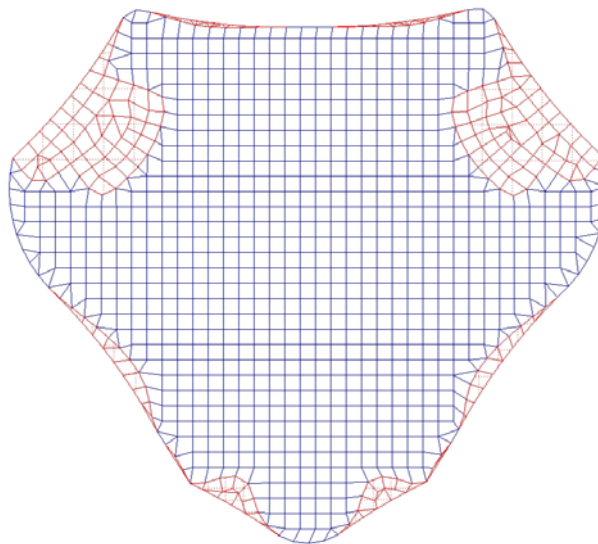


Figure 7.3.3 Blank mesh after smoothing

### 7.3.3 BLANK FITTING

This function enables the user to fit the estimated blank outline by using four straight lines. The dialog box is shown in Figure 7.3.4.

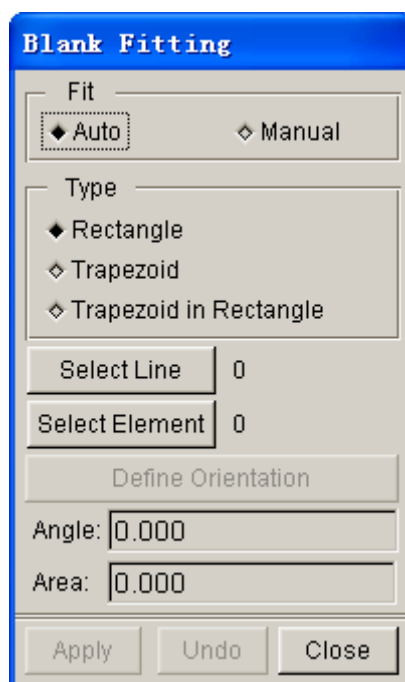
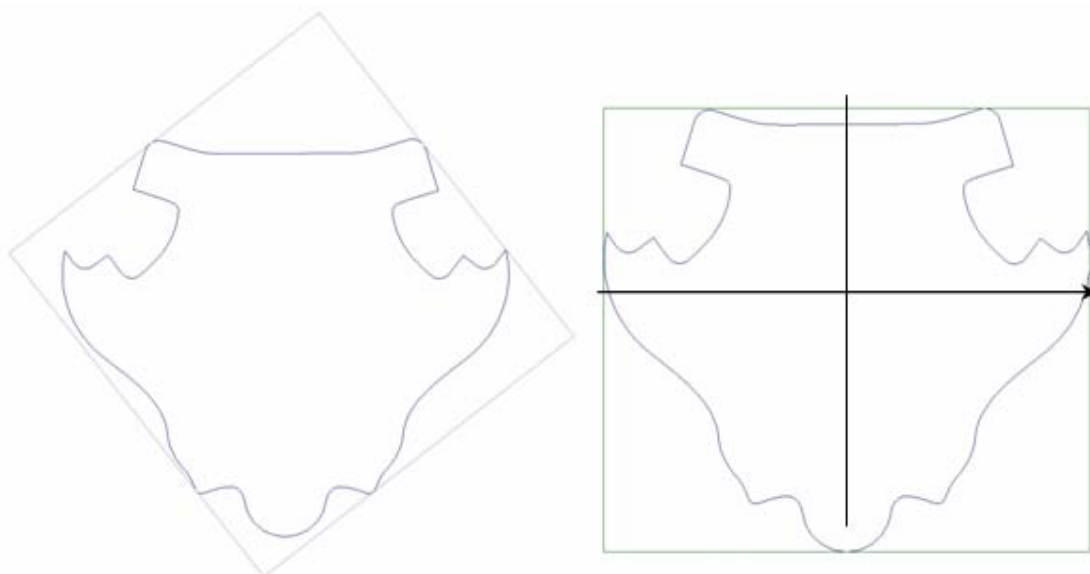


Figure 7.3.4 The Rectangular Fitting dialog box

There are two types of BLANK FITTING: AUTO FIT and MANUAL FIT. If AUTO FIT is selected, the Angle (rotation) and Area (rectangular blank) will be automatically determined. An example of rectangular blank outline is shown in Figure 7.3.5.



a) Auto Fit

b) Manual Fit

Figure 7.3.5 Rectangular blank outline created by Rectangular Fit

To create the rectangular blank outline:

- Select a type of RECTANGULAR FIT (AUTO or MANUAL)

- If MANUAL FIT is toggled, click on DEFINE ORIENTATION to define blank orientation.
  - Click and hold the left mouse button at the cross hair.
  - Move the mouse to rotate the rectangular outline.
  - Stop moving the mouse and release the left mouse button after obtaining the desired orientation.
- Select the estimated blank outline by clicking on the SELECT LINE button. Alternatively, the user can select the blank elements generated by BLANK GENERATION.
- Click APPLY button.

### BLANK FIT TYPE

#### RECTANGLE

If the user selects this blank fitting type, the program fits the blank with a Rectangle as shown in Figure 7.3.6

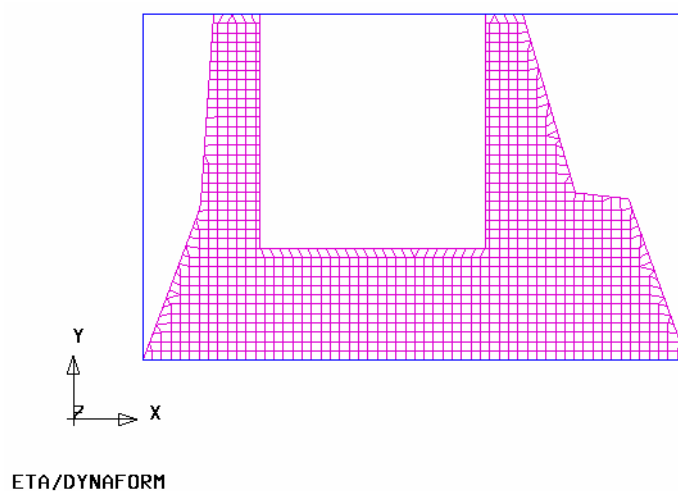


Figure 7.3.6 Rectangle Fit

#### TRAPEZOID FIT

If the user selects this blank fitting type, the program fits the blank with a Trapezoid as shown in Figure 7.3.7.

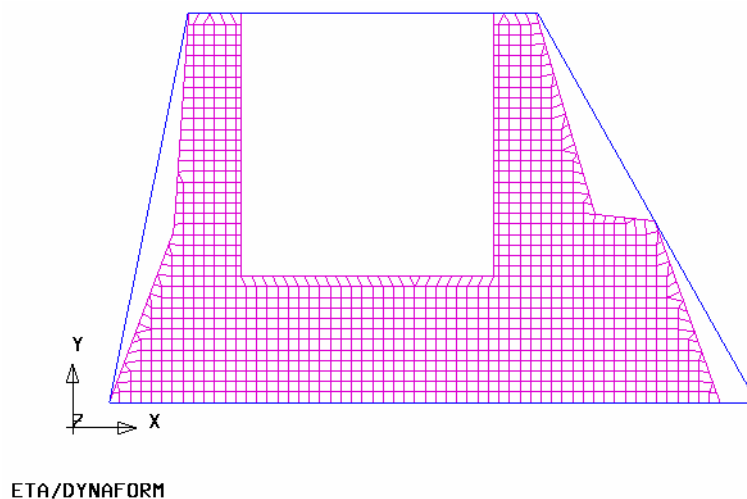


Figure 7.3.7 Trapezoid Fit

**TRAPEZOID IN RECTANGLE**

If the user selects this blank fitting type, the program fits the blank with a Trapezoid in a smallest Rectangle as shown in Figure 7.3.8. The difference between TRAPEZOID IN RECTANGLE and TRAPEZOID is insignificant. However, for some specific examples, the TRAPEZOID IN RECTANGLE type helps to avoid the shape angle.

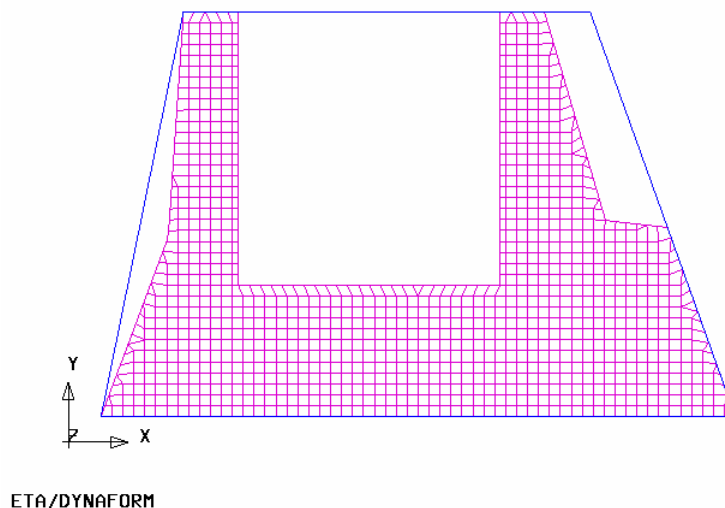


Figure 7.3.8 Trapezoid in Rectangle Fit

**7.3.4 BSE REPORT**

This function enables the user to generate a summary report of the Blank Size Estimation, which is saved as a \*.htm file. The interface of the Blank Size Estimate Report is shown in Figure 7.3.9

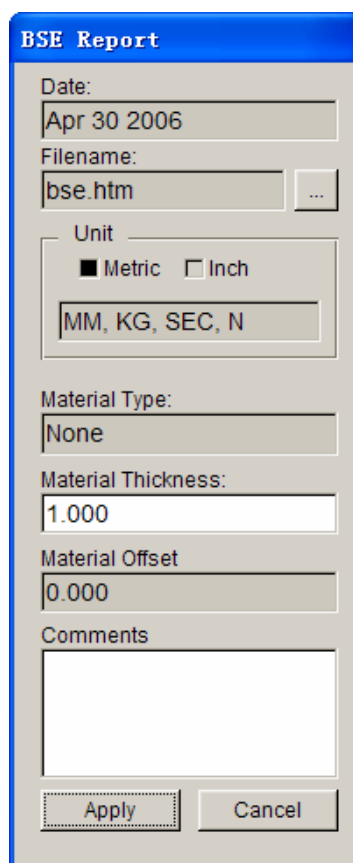


Figure 7.3.9 Blank Evaluation Report

When the Blank Evaluation Report dialog box pops up, there is a highlight frame that appears in the display area. The image displayed inside this frame will be copied as a JPG image to the BSE report .htm file.

**DATE**

Display the date of the blank evaluation report.

**FILE NAME**

Enable the user to assign a file name and the file path to save the current blank evaluation report.

**UNIT**

Display the current unit system in the blank evaluation report.

**MATERIAL TYPE**

Display the name of the selected blank material.

**MATERIAL THICKNESS**

Display the selected material thickness. The user may edit this field.

**MATERIAL OFFSET**

Displays the user defined material offset used for calculations.

**COMMENTS**

Enable the user to write comments about this blank evaluation.

Click on the Apply button to save the current blank evaluation report as the .htm file and then open this file automatically in the default web browser (PC with Windows operating system). A sample BSE report is shown in Figure 7.3.10.

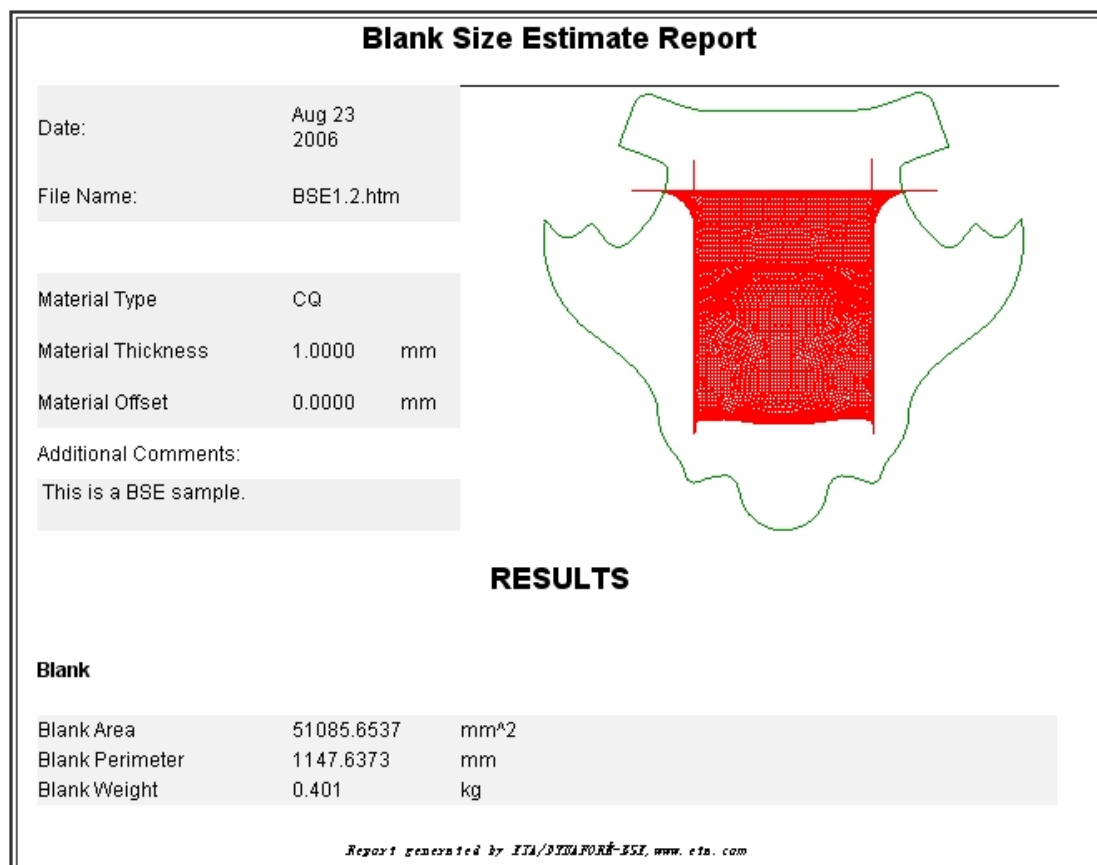


Figure 7.3.10 A sample blank evaluation report

**7.3.5 EXPORT**

This function enables the user to export the blank outlines in various CAD data format.

**7.3.6 BLANK NESTING**

This function enables the user to nest the blank in a strip. This function may be invoked by clicking the **BSE/DEVELOPMENT/BLANK NESTING**. Figure 7.3.11 shows the types of nesting options available in the BLANK NESTING menu.



Figure 7.3.11 Blank Nesting Types

**ONE-UP NESTING** 

This type enables user to generate the nesting result in one row.

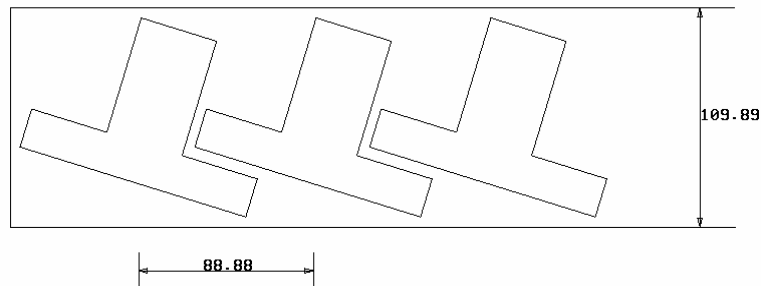


Figure 7.3.12 One-up Nesting

**TWO-UP NESTING** 

This type enables the user to generate the nesting result with two identical blanks with a common orientation.

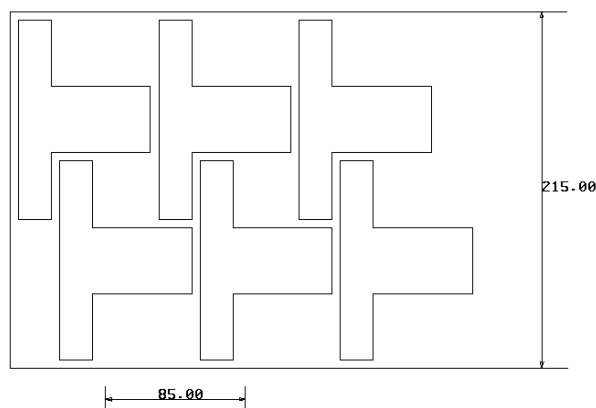


Figure 7.3.13 Two-up Nesting

**TWO-PAIR NESTING** 

This type enables the user to generate the nesting result with a pair of identical blanks with opposite orientation.



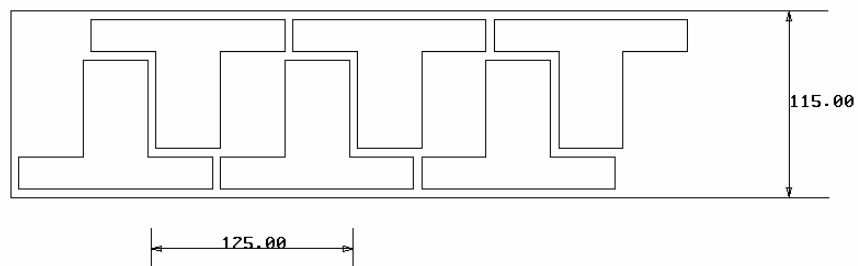


Figure 7.3.14 Two-pair Nesting

### MIRROR NESTING

This type enables the user to generate the nesting result with a single mirrored blank.

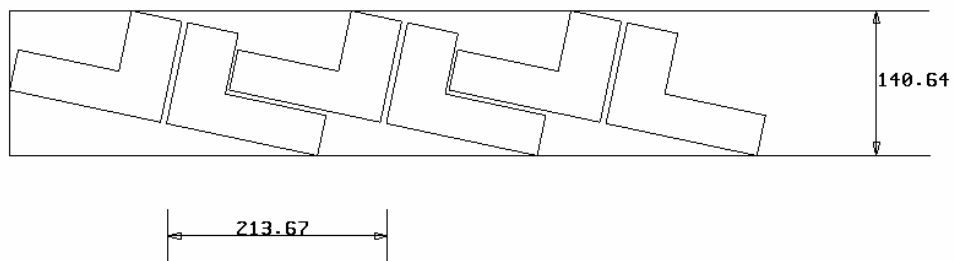


Figure 7.3.15 Mirror Nesting

### MULTI BLANKS NESTING

This type enables the user to generate the nesting result with multiple blanks of different shape.

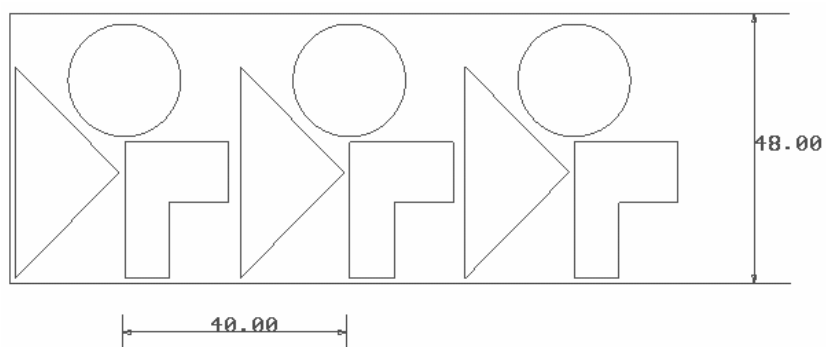


Figure 7.3.16 Multi blanks Nesting

An example of blank nesting is shown in Figure 7.3.12.to Figure 7.3.16. A detailed description of each function is given in the following section.

### 7.3.6.1 ONE-UP NESTING

This function enables the user to generate the nesting result in one row. The dialog box is shown in Figure 7.3.17.

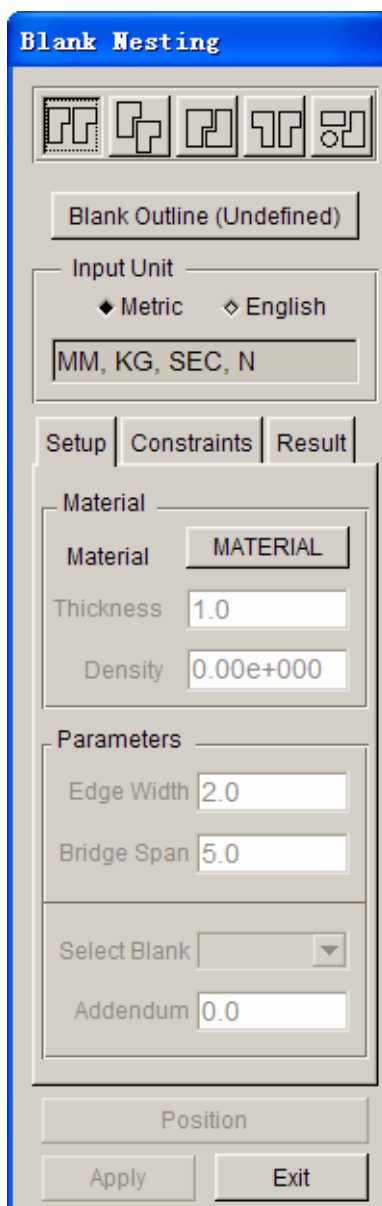


Figure 7.3.17 One-up Nesting dialog box

- **Blank Outline (Undefined)**

This parameter is used to define the Blank Outline. Click on the **Blank Outline (Undefined)** button in Figure 7.3.17, the program displays the SELECT LINE dialog box and prompt the user to select lines. After the Blank Outline is selected, the button name is changed to **Blank Outline**.

#### INPUT UNIT (toggle on)

##### **Metric**

This option enables the user to define the Nesting setup parameters in metric unit.

**English**

This option enables the user to define the Nesting setup parameters in inch unit.

**SETUP**

In SETUP tab, the user can define the base parameters for nesting of blanks such as defining the material of the blank and input of the Edge Width, Bridge Span and Addendum size.

**MATERIAL**

The user can define the material of blank using two methods. In first method, the user can click on the MATERIAL button to open the Material definition dialog box. The user may choose to create a NEW material model or IMPORT a material. Once the material is defined, the density is displayed in the lower window, and part thickness can be defined. Another method is by entering the THICKNESS and DENSITY directly in the corresponding data fields.

**Thickness**

This box enables the user to enter the thickness of blank.

**Density**

This box enables the user to enter the density of part material.

**PARAMETERS**

The user may define parameters used to control the gaps of blank nesting such as Edge Width, Bridge Span and Addendum. Figure 7.3.18 shows these parameters related to the blank outline.

**Edge Width**

Edge Width allows the user to define the gap between blank outline and strip edge.

**Bridge Span**

Bridge Span allows the user to define the gap between parts.

**Select Blank**

This option enables the user to choose between multiple blanks to assign different or common addendum sizes. Please note that this option is only available for nesting options with more than one blank outline.

**Addendum**

Addendum allows the user to expand the selected blank outline.

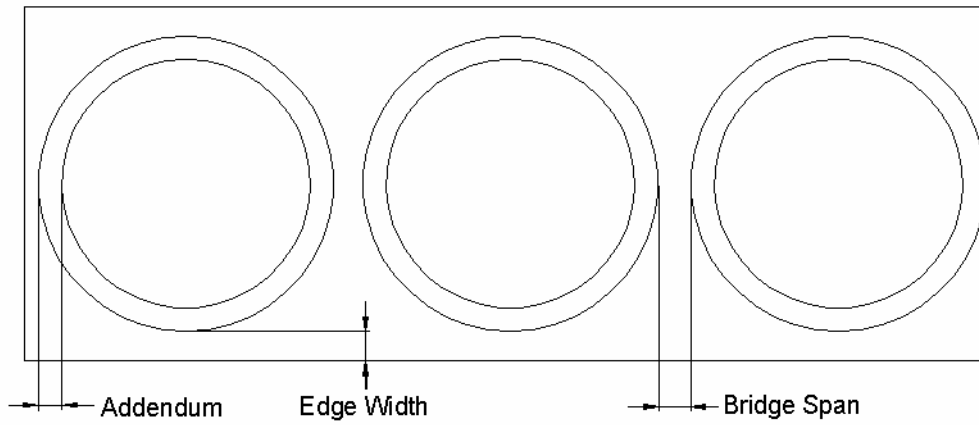


Figure 7.3.18 Edge Width, Bridge Span and Addendum parameters

### CONSTRAINTS

This function enables the user to constrain the dimension of the strip and the angle of blank relative to the strip direction. The tab is illustrated in Figure 7.3.19.

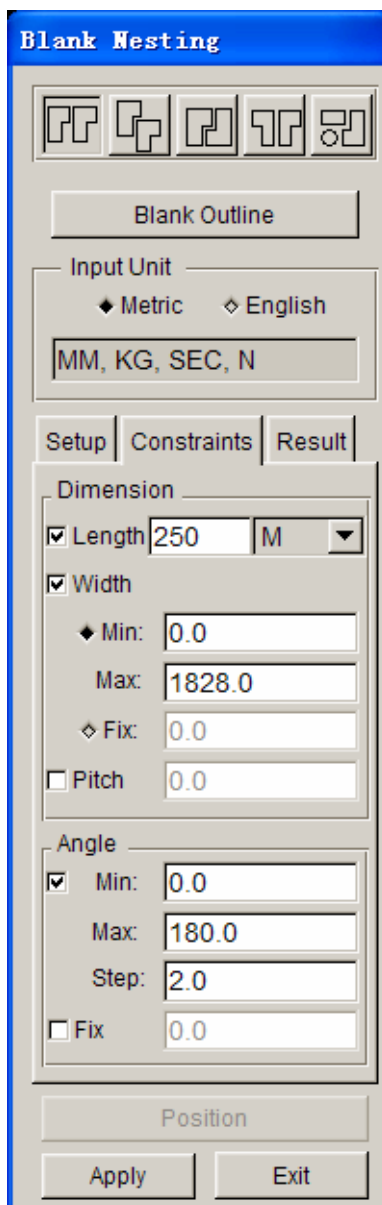


Figure 7.3.19 Constraints

## DIMENSION OPTION

### LENGTH

Fix Length is used to define the total strip or sheet length.

### WIDTH

Width is used to constrain the width of the strip or sheet.

- The width range is used as the width constraint. The user defines the allowable range of the blank width (Min and Max). eta/DYNAFORM Nesting calculates a series of results according to the defined range of the width.

- Fix width is used to define a width for the strip or sheet.

### PITCH

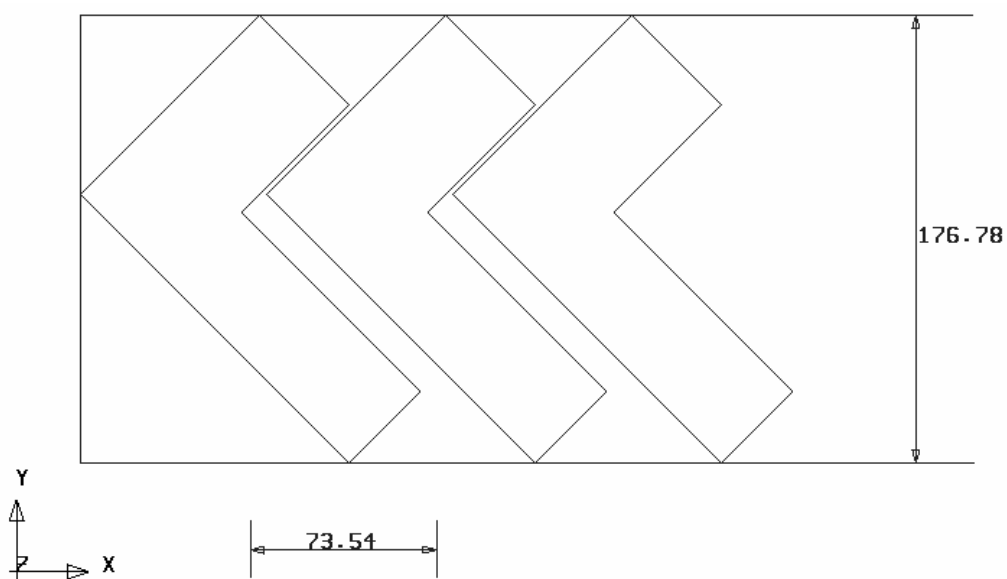
The Fix Pitch can constrain the pitch in nesting.

### ANGLE OPTION

- The angle range is used as the angle constraint. The user defines the allowable range of the blank rotation (Min and Max). eta/DYNAFORM Nesting calculates a series of results according to the defined range of the rotation angle.
- Fix angle used to specifically define a rotation angle for the blank.

### APPLY

Click on the **Apply** button as illustrated in Figure 7.3.19, to run the blank nesting and to show the optimal result for material usage, as illustrated in Figure 7.3.20.



ETA/DYNAFORM

Figure 7.3.20 The Result of One-up Nesting

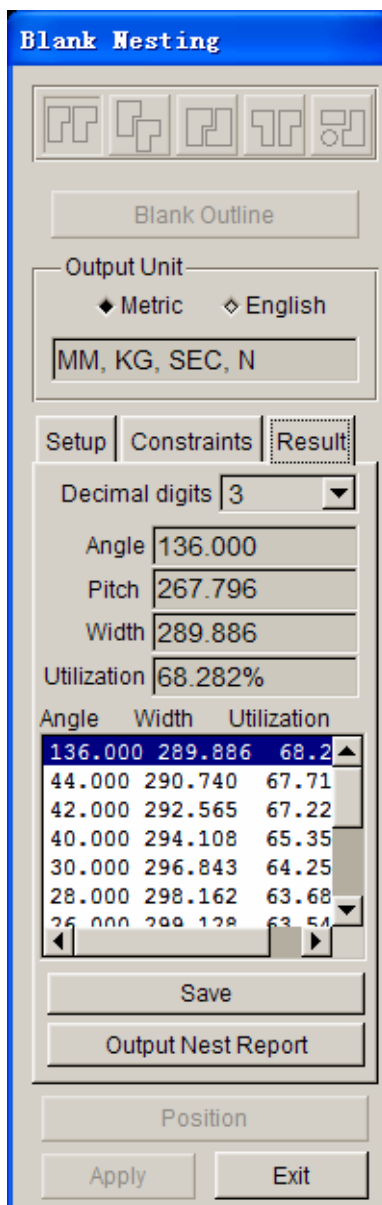


Figure 7.3.21 Result of Blank Nesting

The program offers the user a series of available blank nesting results that meet the given constraints. The user may select the number of decimal digits to display by selecting the number in the Decimal digits combo box. The user may click any result from the result list window and the corresponding constraints will be displayed in the text boxes above the window. As shown in Figure 7.3.21, there are five results for the user to select from. The user may further reduce the constraint range according to the first results to get more detailed nesting schemes.

**NOTE:** The user may not change the nesting type or the unit system when the **RESULT** table is the current tab.

- **Output Nest Report**

This function enables the user to output the nesting result as a HTML file. The interface of the Output Nesting Report is shown in Figure 7.3.22.

**Nest Report**

Date: Jul 07 2006

Filename: nest.htm

Blank Outline Color

Output Unit: MM, KG, SEC, N

Production Volume / Per Coil: Production Volume

Production Volume: 100000

Coil Length: 250.000 M

Base Material Cost: 2.0 \$/kg

Extra Material Cost: 0.400 \$/kg

Scrap Value Cost: 0.2 \$/kg

Consumables Cost: 0.350 \$/blank

Comments: A sample of Nesting report

Apply Cancel

Figure 7.3.22 The Output Nesting Report dialog box

When the Output Nesting Report dialog box pops up, there is a highlighted picture frame appears in the display area. The image displayed inside this frame will be copied to the nesting report HTML file.

#### **DATE**

Display the date of the nesting report.

#### **FILE NAME**

Enable the user to assign a file name (.htm) and the file path to save the current nesting report.

#### **BLANK OUTLINE COLOR**



Enable the user to change the color of the blank outline.

**UNIT**

Display the current unit system in the nesting report.

**PRODUCTION VOLUME / PER COIL**

Enable the user to select PRODUCTION VOLUME or PER COIL for cost calculation.

**PRODUCTION VOLUME (×1000)**

Enable the user to define the production volume.

**COIL LENGTH**

Display the defined coil length, defined in Constraints.

**BASE MATERIAL COST**

Enable the user to define the base material cost.

**EXTRA MATERIAL COST**

Enable the user to define the extra material cost.

**SCRAP VALUE COST**

Enable the user to define the scrap value cost.

**CONSUMABLES COST**

Enable the user to define the consumable cost.

**COMMENTS**

Enable the user to write comments about this nesting in the nesting report.

Click on the Apply button to save the current nesting report as.htm file and then open this file automatically in the default web browser (PC with Windows operating system). A sample nesting report is shown in Figure 7.3.23.

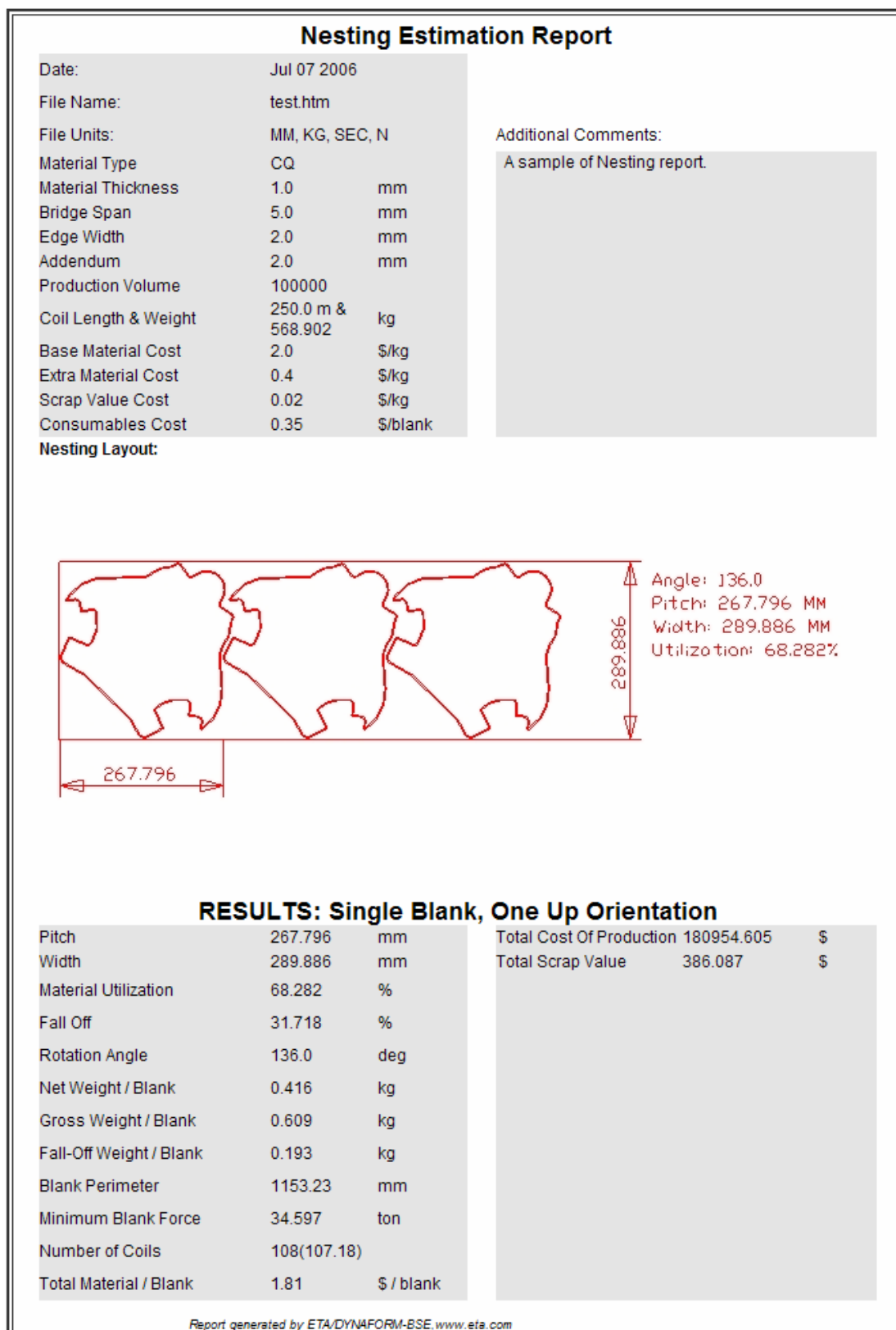


Figure 7.3.23 A typical nesting report

### 7.3.6.2 TWO-UP NESTING

This function enables the user to generate the nesting result of two identical blanks with common orientation. The setting of Setup and Constraint in this function is similar to that in the One-up Nesting. Refer to Section 7.3.5.1 for description of commonly used parameters.

The Position option is activated after the blank outline is selected. This option enables the user to adjust the relative position between two blanks. The user may use the automatic tools to optimize the blank orientations. Else, the user may manually select a blank outline, the desired X- or Y-axis translations, and/or Z-axis rotations. Click on the Position button to open the Blank Position dialog box as shown in Figure 7.3.24.

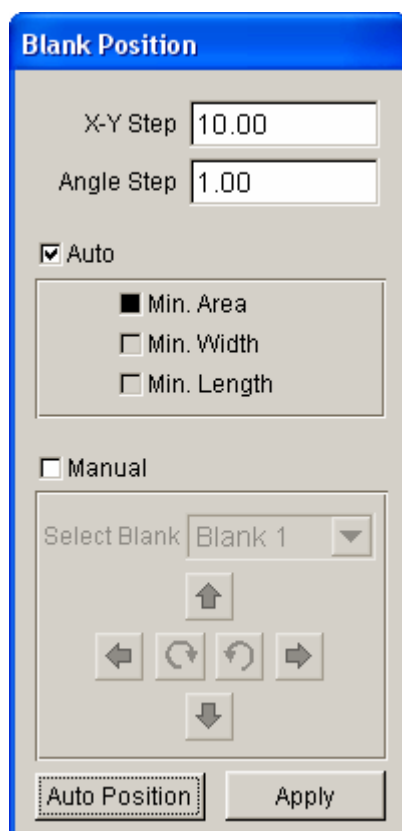


Figure 7.3.24 Auto Position dialog box

### X-Y STEP

This function allows user to set the incremental value in X and Y direction for transformation. This increment is useful both in Auto and Manual Nesting. If the user adjusts the position of the blank through the Manual method for transformation, the blank is moved along the X/Y direction by the incremental value from each click of the translation button. If the Auto method is selected, the program calculates a series of results according to the step along the X/Y direction. The default X and Y step is 10.

### ANGLE STEP

This is the angle increment value for blank rotation. If the user adjusts the position of the blank through Manual method for rotation, the blank is rotated by the incremental angle from each click of the

rotation button. If the Auto method is selected, the program calculates a series of results according to the angle increment. The default angle step is 10 degree.

## AUTO

This function enables the user to position the nesting blanks automatically through the selected optimizing scheme.

- **Min. Area**

The program will position the nesting blanks to minimize the surface area of the strip.

- **Min. Width**

The program will position the nesting blanks to minimize the width of the strip.

- **Min. Length**

The program will position the nesting blanks to minimize the length of progression.

## MANUAL

This function enables the user to position the blanks manually through translation and/or rotation.

### SELECT BLANK

This option enables the user to select the desired blank to position when there are more than one blank defined for nesting. The selected blank is highlighted in the display area.

### TRANSFORMATION



Move the selected blank outline incremental distance in positive X-axis direction



Move the selected blank outline incremental distance in negative X-axis direction



Move the selected blank outline incremental distance in positive Y-axis direction



Move the selected blank outline incremental distance in negative Y-axis direction

### ROTATION



Rotate the selected blank outline clockwise by the incremental angle.



Rotate the selected blank outline counter-clockwise by the incremental angle.

**AUTO POSITION**--Enables the user to obtain the positioned result automatically.

**APPLY**--The program accepts the positioning result and exit the position dialog box.

### **7.3.6.3 TWO-PAIR NESTING**

This function enables the user to generate the nesting result with two identical blanks with opposing orientation. The setting of Setup and Constraint in this function is similar to that in the Two-Up Nesting. Refer to Sections 7.3.6.1 and 7.3.6.2 for all relevant parameters.

### **7.3.6.4 MIRROR NESTING**

This function enables the user to generate the nesting result with mirroring the original blank. The mirrored blank is symmetrical about X or Y direction. The setting of Setup and Constraint in this function is similar with that in the Two-Pair Nesting. Refer to Sections 7.3.6.1. and 7.3.6.2 for description about all relevant parameters.

### **7.3.6.5 MULTI BLANKS NESTING**

This function enables the user to generate the nesting result with more than one different blank outline. During the definition of the Blank Outline, the user should click OK button or the middle button to accept each profile selection after selecting each blank outline. When all the blank outlines are selected, the user can click OK button or the middle button directly to exit the SELECT LINE dialog box.

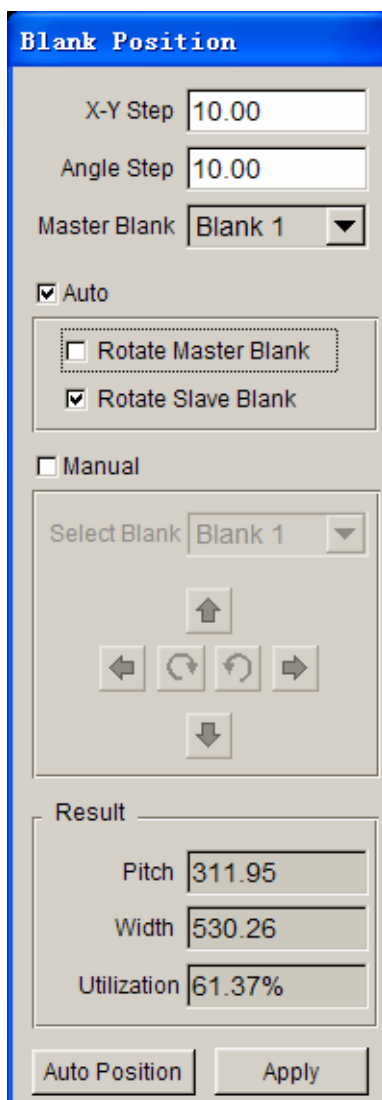


Figure 7.3.25 Blank Position dialog box

**MASTER BLANK**

This option enables the user to select the master blank. This option is active only when the Multi-blanks nesting type is used.

*NOTE: Since the program uses the defined X-Y step and Angle step to calculate the nesting result in Auto Nesting, the user may use the default value to calculate to void the long computing time. Then, reduces those values gradually to get more accurate result.*

**AUTO**

This function enables the user to position the blanks automatically through the selected optimizing scheme.

**ROTATE MASTER BLANK**

This option allows the user to control whether the master blank is rotated during auto nesting.

**ROTATE SLAVE BLANK**

This option allows the user to control whether the slave blanks are rotated during auto nesting. This option is only activated for multiple blank nesting.

## Chapter 8

# SETUP

The functions provided in the SETUP user graphic interface (GUI) help the user to quickly setup a forming simulation. The SETUP menus include two types of setup: One is the traditional QUICK SETUP consists of several standard forming operations as shown in Figure 8.1; another is the new AUTOSETUP.

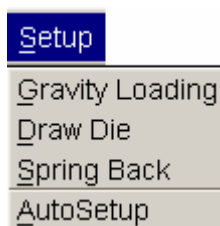


Figure 8.1 The Setup Menu

### 8.1 QUICK SETUP (QS)

The standard-forming processes supported by QS are the following:

#### GRAVITY LOADING

#### DRAW DIE

- Crash Form
- Inverted Draw (Single Action)
- Toggle Draw (Double Action)
- Four Piece Draw (Double Action Stretch Draw)

#### SPRINGBACK

The QS GUI is designed and embedded in eta/DYNAFORM as a streamlined, user-friendly and fully automated interface. The QS interface offers contact offset algorithm in the stamping simulation. This capability eliminates the potential mesh generation problem in geometrical mesh offset. Compared to a simulation solution using geometrical mesh offset, it provides a quicker solution for a stamping simulation without significantly compromising accuracy.

The following table outlines the major differences among the QUICK SETUP, AUTO SETUP and the traditional eta/DYNAFORM stamping simulation setup procedure.



TRADITIONAL SETUP	QUICK SETUP	AUTO SETUP
Manual interface can duplicate any tooling configuration: pads, multiple tools, etc.	Automated interface limits flexibility.	Provides built-in templates that enable user to setup all types of operation.
Requires longer setup time.	Reduces modeling setup time.	Reduces setup time and reduces the possibility of user mistake.
Manual definition of tool travel curves.	Automated tool travel curves.	Automated tool travel curves and user defined curve.
Geometrical Offset.	Contact Offset.	Supports both Contact Offset and Geometrical Offset.

Most of the QS GUI contains following options:

**AUTO ASSIGN** allows the user to assign tools that have a standard part name, such as BLANK, DIE, BINDER and PUNCH.

**CONSTRAINT** allows the user to define SPC (single point constraint) for symmetric or other boundary conditions.

**ADVANCED** allows the user to change some default parameters that relate to QS settings.

**HELP** provides useful tips for stamping simulation.

**APPLY** activates the program to automatically copy the mating tool, calculate contact offset parameters and define the travel curves as well as binder force curve.

**RESET** allows the user to recover the stage before creating the mating tool.

**PREVIEW** enables the user to check tool movement.

**SUBMIT JOB** enables the user to run the analysis via the ANALYSIS menu (refer to Section 14.1).

**EXIT** allows the user to exit the QS interface.

A typical QS GUI is shown in Figure 8.1.1.

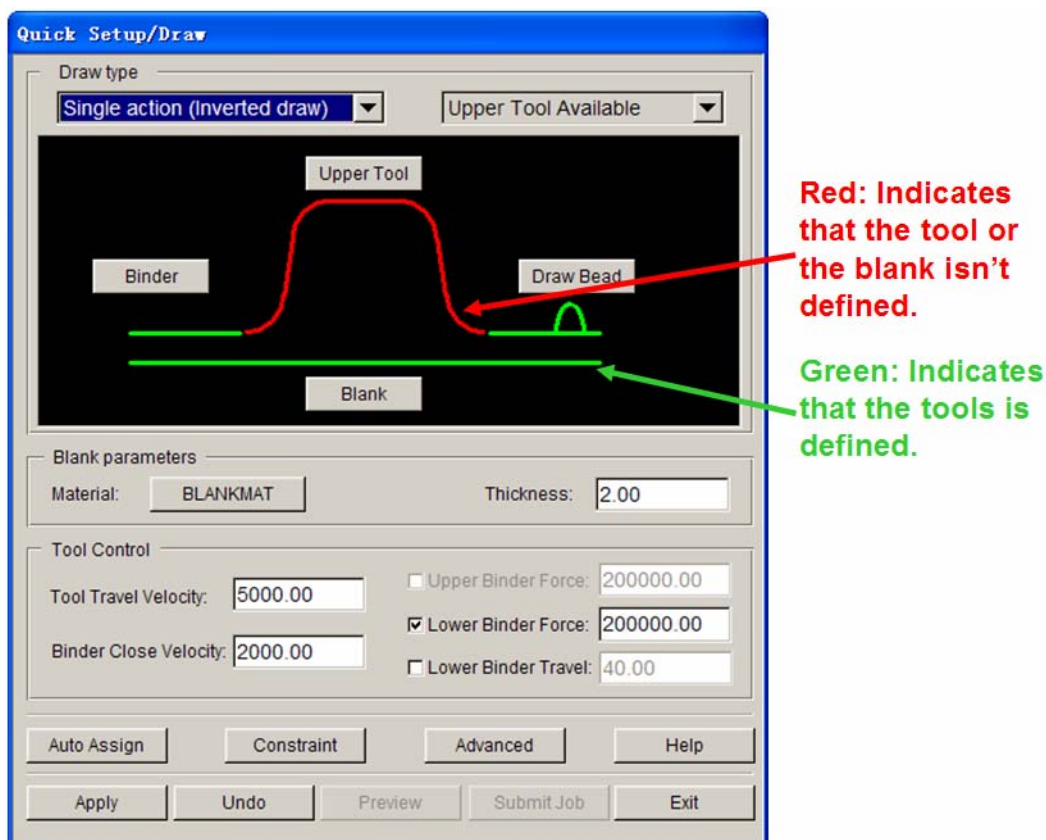


Figure 8.1.1 A Typical Quick Setup Graphic User Interface

### 8.1.1 GRAVITY LOADING

The QUIK SETUP/GRAVITY LOADING GUI leads the user through the setup procedure for gravity loading simulation. A typical GRAVITY LOADING GUI consists of DRAW TYPE, TOOL DEFINITION (BLANK, BINDER and/or LOWER TOOL), BLANK PARAMETERS, SOLVE OPTION and CONTROL as shown in Figure 8.1.2.

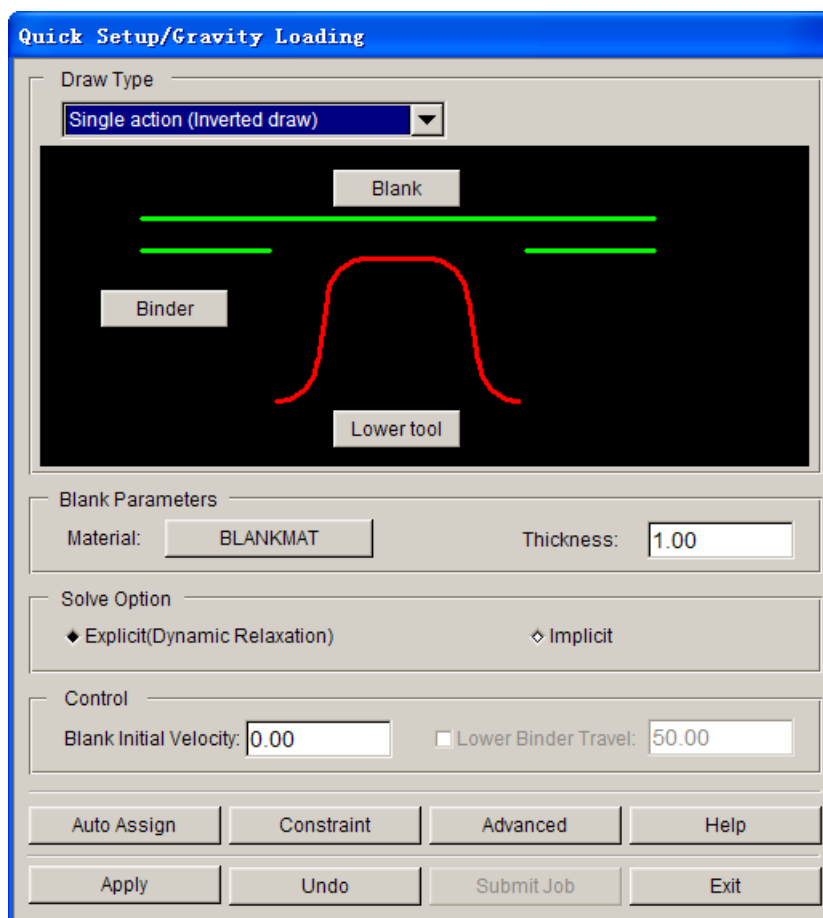


Figure 8.1.2 A Typical Gravity Loading GUI for Single Action Draw Operation

### **DRAW TYPE**

Enable the user to select one of the four standard draw operations provided in QS GUI.

### **TOOL DEFINITION GUI**

Define the tool and blank for a gravity loading simulation.

### **BLANK PARAMETERS**

Define blank material and properties.

### **SOLVE OPTIONS**

The user may select either the EXPLICIT or IMPLICIT method for a gravity loading simulation. The EXPLICIT method requires a longer simulation time than the IMPLICIT method, but it provides good reliability. Although the IMPLICIT method yields an accurate solution, it is difficult to achieve converged non-linear equilibrium iterations when using this method. Therefore, the EXPLICIT method is recommended for a gravity loading simulation.

**CONTROL**

Define the initial velocity of blank.

**8.1.1.1 TOOL DEFINITION GUI**

The tool definition GUI allows the user to assign the tooling and blank mesh required for setting up a particular stamping simulation. For a SINGLE ACTION GRAVITY LOADING simulation (as shown in Figure 8.1.2), the tool definition GUI consists of BLANK, BINDER and LOWER TOOL.

By clicking on either the BLANK, BINDER or LOWER TOOL button, the dialog boxes as shown in Figure 8.1.3 will prompt the user to select the method of defining blank or tooling.

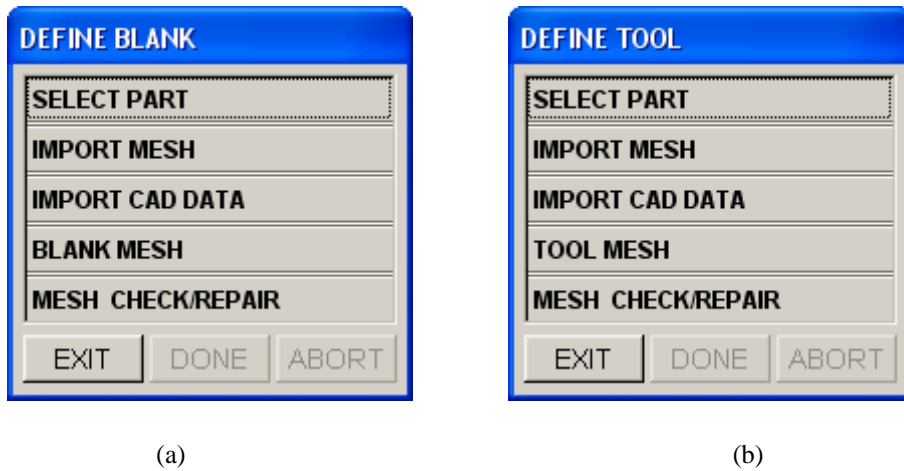


Figure 8.1.3: The Tool Definition dialog box (a) Define Blank (b) Define Tool

If the blank mesh is available in a saved NASTRAN or DYNA file, the user can click the IMPORT MESH function to read in the blank mesh. The QS interface will automatically assign the part (blank mesh) as BLANK. Alternatively, the user can read in the blank geometry (in CAD format) by selecting IMPORT CAD DATA function. The BLANK MESH function (as shown in Figure 8.1.3a) is then used to mesh the blank geometry. This function is similar to BLANK GENERATOR described in Section 9.5. After completing the mesh generation, the QS interface will automatically assign the part (blank mesh) as BLANK.

The TOOL MESH function (as shown in Figure 8.1.3b) meshes the tooling geometry. For detailed information about this function, refer to Section 5.3.4. The QS interface will also automatically assign the tool mesh as either BINDER or LOWER TOOL after the tool geometry is meshed.

The SELECT PART function (Figure 8.1.3a) allows the user to select the proper parts from the SELECT PART dialog box (Figure 8.1.4) and assign them as BLANK, LOWER TOOL and BINDER in the QS GUI. By clicking on SELECT PART function, the DEFINE BLANK dialog box is displayed, and the user can continue by adding the corresponding parts as BLANK, LOWER TOOL and BINDER. The SELECT PART function in the DEFINE TOOL dialog box (Figure 8.1.3b) is displayed. The DEFINE TOOL dialog box as shown in Figure 8.1.5.

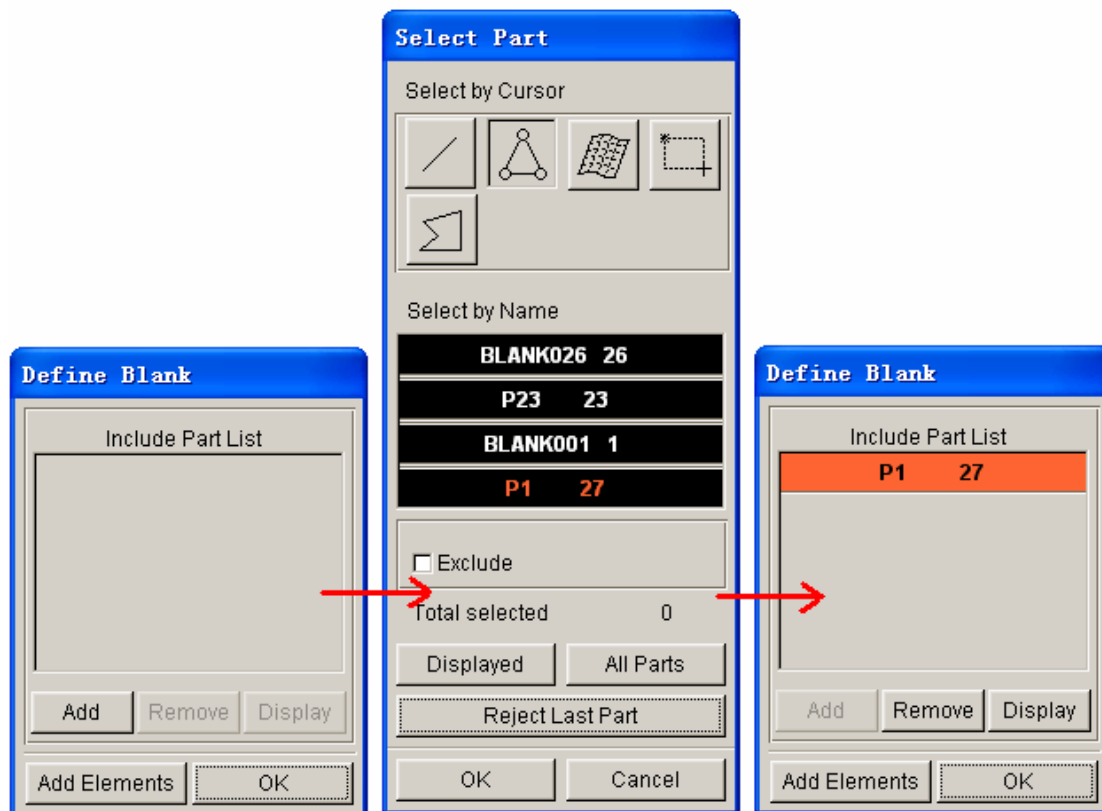


Figure 8.1.4 A Schematic Illustration Showing the SELECT PART Operation for Blank Definition in QS

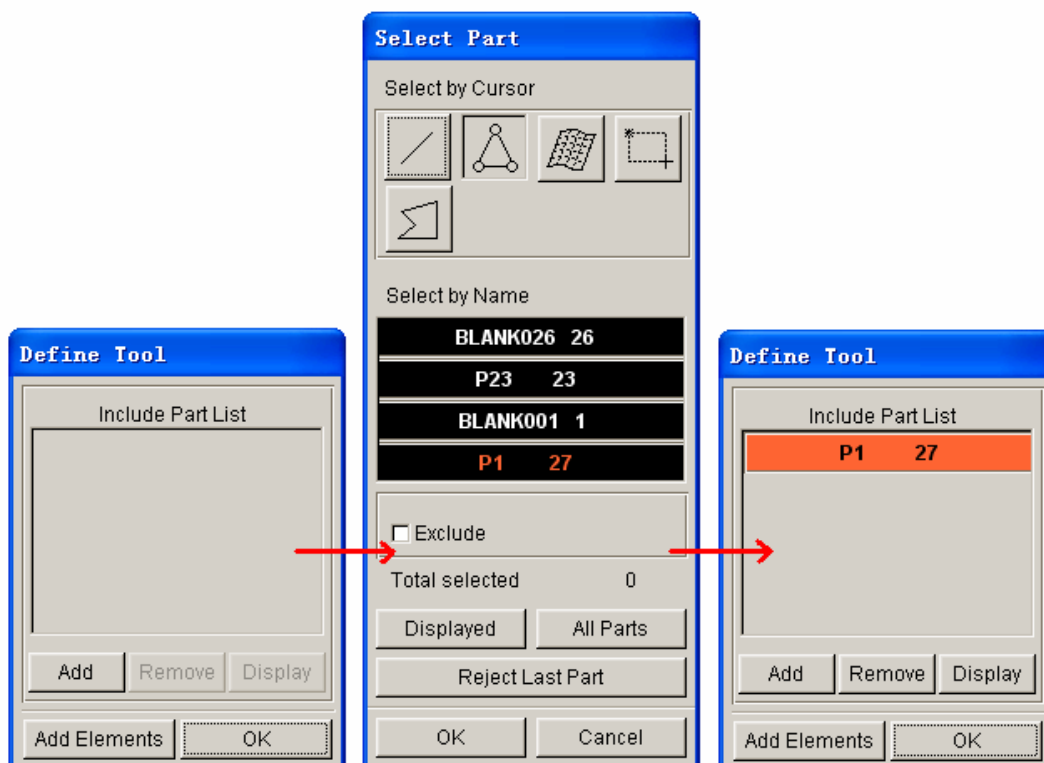


Figure 8.1.5 A Schematic Illustration Showing the SELECT PART Operation to Define the Lower Tool in QS

### 8.1.1.2 BLANK PARAMETERS

The user may define the blank material by selecting the NONE button as shown in Figure 8.1.2. The program will display the MATERIAL dialog box as shown in Figure 8.1.6. For detailed information about defining the blank material, refer to Section 10.9. The NONE button is disabled if no blank is defined in the tool GUI.

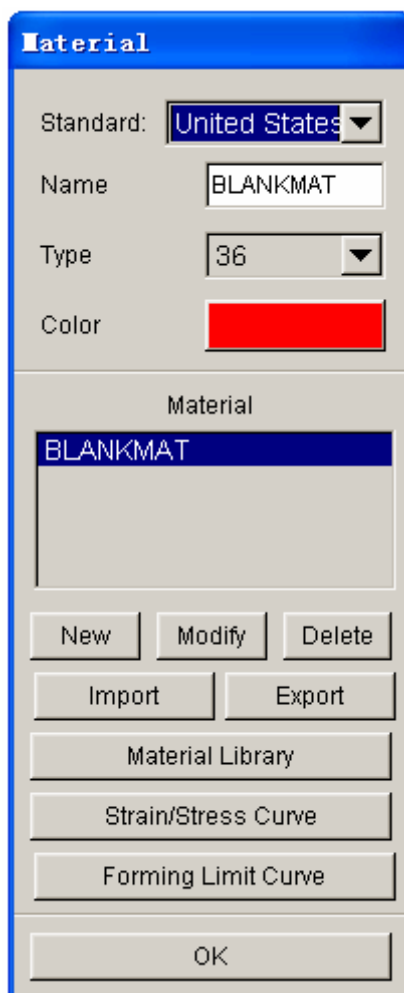


Figure 8.1.6 The Material dialog box

### 8.1.1.3 QUICK SETUP/GRAVITY LOADING PROCEDURE

Follow the procedures listed below to set up the SINGLE ACTION gravity loading simulation.

1. Select QUICK SETUP/GRAVITY LOADING.
2. Select DRAW TYPE.
3. Click on the BLANK/BINDER/LOWER TOOL button from the QS GUI to define/assign blank/binder/die mesh. If the process doesn't involve punch support, the user can ignore the lower tool definition.

4. Define the blank material. If a blank material is not defined, NONE will be shown on the button. Click the NONE button to select a blank material from the material library or create a user defined material.
5. Key in the blank thickness. The default blank thickness is set as 1.00 (mm).
6. Select the solver option.
7. Click APPLY to auto-position the tools and blank.
8. Click SUBMIT JOB to submit the analysis. Refer to Section 14.1 for more details.

## 8.1.2 DRAW

The QUICK SETUP/DRAW GUI leads the user through the setup procedure for draw simulations. A typical DRAW GUI consists of DRAW TYPE, TOOL DEFINITION (BLANK, BINDER, LOWER TOOL and DRAW BEAD), BLANK PARAMETERS, and TOOL CONTROL, as shown in Figure 8.1.7.

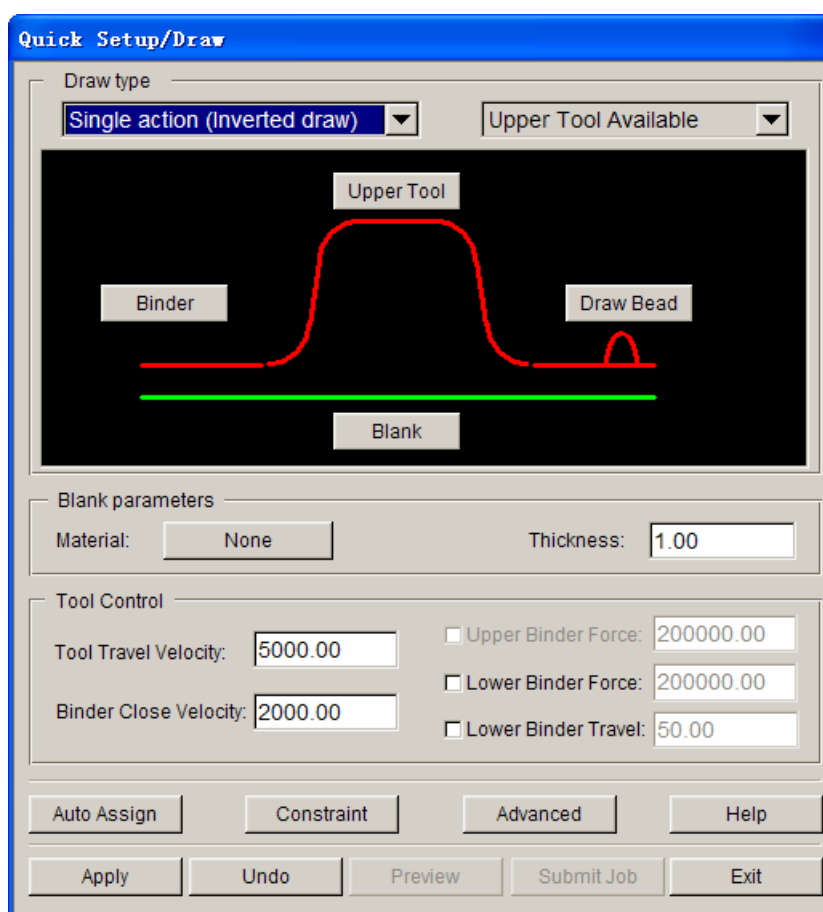


Figure 8.1.7 A Typical Draw GUI for Single Action Draw Operation

**DRAW TYPE**

Enable the user to select one of the four standard draw operations provided in the QS GUI. In addition, the user may select either lower or upper tool in the setup depending on the type of tool surfaces (top or bottom).

**TOOL DEFINITION GUI**

Define the tool, blank and drawbead for draw simulation.

**BLANK PARAMETERS**

Define blank material and properties. Refer to Section 8.1.2.

**TOOL CONTROL**

Enable the user to define the motion of the tool.

**ADVANCED**

Allow the user to edit some of the default QS settings.

**8.1.2.1 TOOL DEFINITION GUI**

The tool definition GUI allows the user to assign the tooling and blank mesh required for setting up a particular draw simulation. Refer to Section 8.1.1.

**8.1.2.2 TOOL CONTROL**

As shown in Figure 8.1.8, the tool control interface is utilized to set primary travel velocity and binder force. The user may activate the binder force control selecting the BINDER FORCE check box.

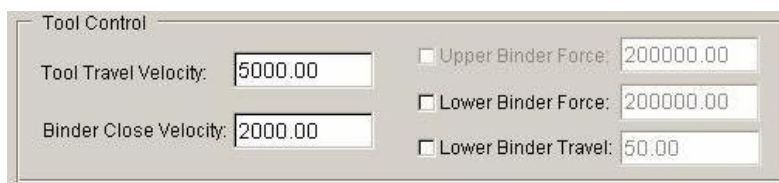


Figure 8.1.8 The Tool Control Interface

Depending on the draw type, the tool velocity and force are applied according to the following control scheme:

**CRASH FORM**

Upper tool with defined tool velocity.



<b>INVERTED DRAW (SINGLE ACTION)</b>	Upper tool is controlled by velocity. Binder is controlled by either velocity or force.
<b>TOGGLE DRAW (DOUBLE ACTION)</b>	Upper tool is controlled by velocity. Binder is controlled by velocity or force.
<b>FOUR PIECE DRAW</b>	Upper tool is controlled by velocity. Upper binder is controlled by velocity or force. Lower binder is controlled by velocity or force.

If the default unit is selected, the unit of velocity and force are in **mm** and **Newton**, respectively.

**NOTE:**

- 1. In the QS interface, all force controls are implemented after the binder is closed using velocity control. That is, the velocity control is switched to force after binder closing.*
- 2. In the QS interface, the force applied to the lower binder (in FOUR-PIECE DRAW) is controlled by velocity or force with rigid body control. Rigid body control will prevent the lower binder from moving up before the upper and lower binders clamp the blank.*

For example (INVERTED DRAW):

If the user needs to control the punch with velocity and the binder with force control:

- Provide punch travel velocity (default is 5000mm/s).
- Provide binder velocity (default is 2000 mm/s).
- Provide binder force (default is 200,000 Newton).

### 8.1.2.3 ADVANCED

This feature allows the user to edit the default QUICK SETUP settings, including the gap, load curve, output frames and drawbead control. The dialog box is shown in Figure 8.1.9.

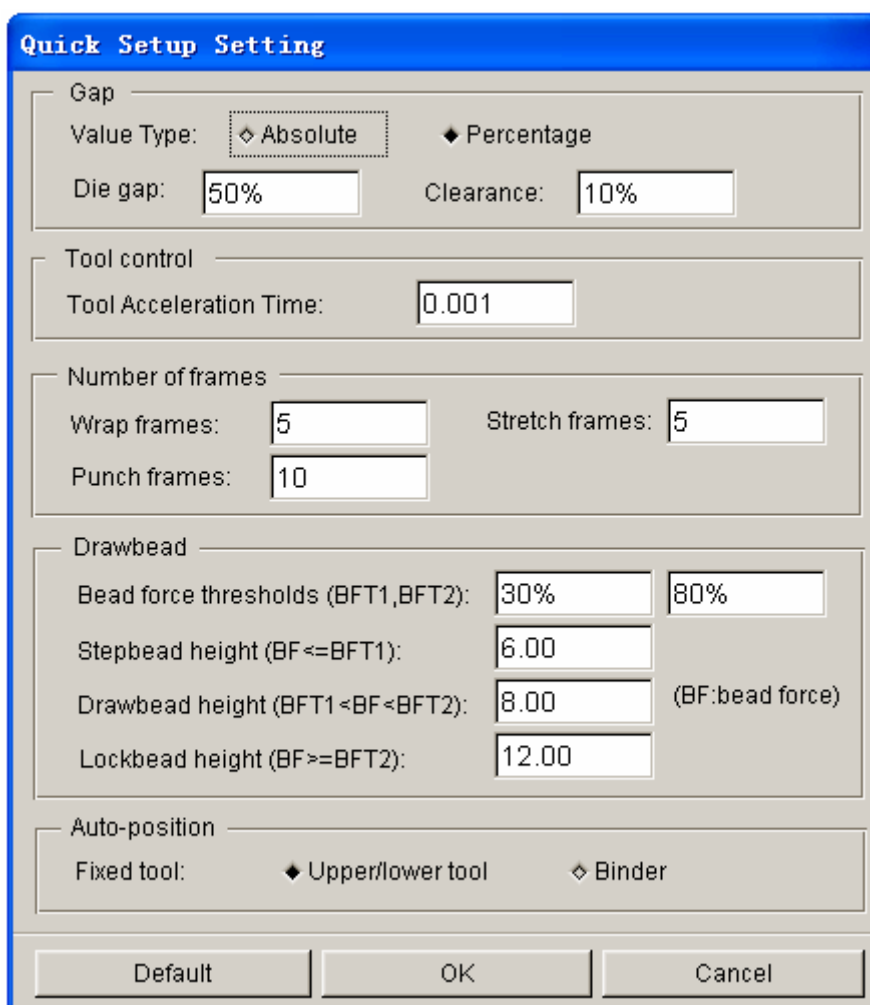


Figure 8.1.9 Quick Setup Setting dialog box

The GAP CONTROL allows the user to set the tooling gap and clearance between the tooling and blank. The user can select either an absolute or percentage value. The default setting for die gap and clearance is 0.10 (110% of blank thickness) and 0.50 (50% of blank thickness). For example, if the blank thickness is 1.00 mm, the tooling gap and clearance is 1.10 mm and 0.50 mm, respectively.

The load curve setting enables the user to define the slopes of the velocity load curve and lower binder travel limitation of four-piece draw simulation.

The user may define the output frames of each stage. The default settings for wrap, stretch and punch frames are 5, 5 and 10, respectively.

The drawbead control enables the user to define drawbead conveniently. For more information about the Drawbead refer to the Section 10.4.

AUTO-POSITION enables the user to set the fixed tool after the user clicks the APPLY button for AUTO-POSITION operation etc.

#### **8.1.2.4 The QUICKSETUP/DRAW PROCEDURE**

The following is the procedure for setting up a draw simulation using QS/DRAW:

1. Select draw type and tooling options.
2. Define tooling. Refer to Section 8.1.1.1.
3. Define blank parameters. Refer to Section 8.1.1.2.
4. Define tool motion. Refer to Section 8.1.2.2.
5. Define drawbeads. Refer to Section 8.1.2.1.
6. Apply symmetry condition. The user can define symmetry boundary conditions by clicking on CONSTRAINT. Refer to Section 5.6.2 for detailed information about SPC options.
7. Select APPLY and SUBMIT JOB. After selecting APPLY, eta/DYNAFORM will create the matching tool, automatically adjust tooling position, generate travel curves and offset the contact interface. The SUBMIT JOB button allows the user to run the simulation via the ANALYSIS dialog box (refer to Section 14.1).

### 8.1.3 SPRINGBACK

The QS/SPRINGBACK GUI (as shown in Figure 8.1.10) allows the user to easily setup a springback analysis.

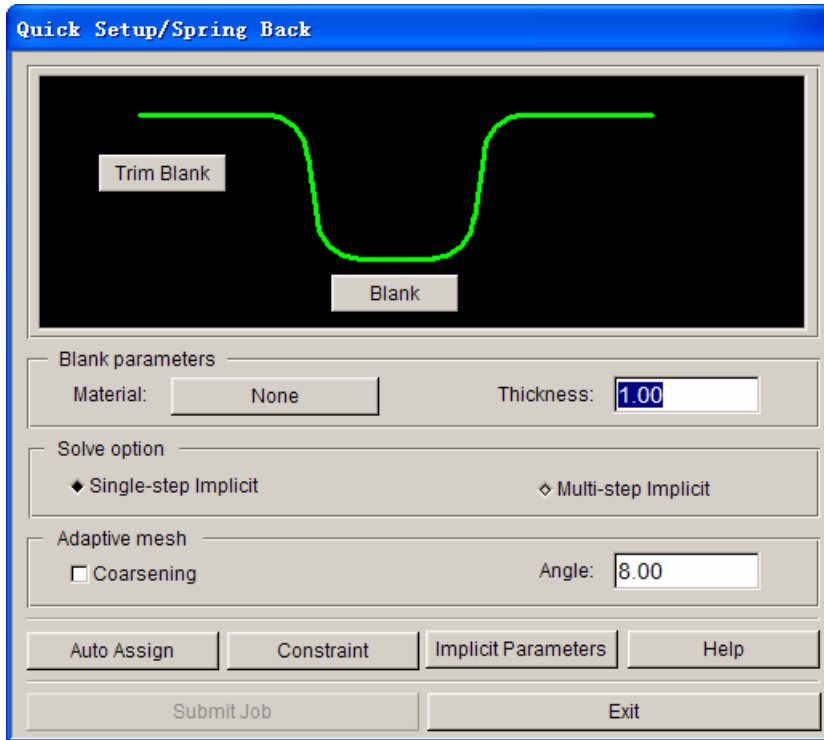


Figure 8.1.10 The QS Springback GUI

#### TOOL DEFINITION GUI

Allow the user to define a deformed blank. Selecting the TRIM BLANK button to carry out the trim operation, the user has to read in the trim curve. By default, the deformed blank is assigned automatically as BLANK after the file (in DYNAIN format) is read into the eta/DYNAFORM database.

#### BLANK PARAMETERS

Define blank material and properties.

#### CONTROL

The user can either select the single or multi-step implicit method. The single-step implicit method is the default setting. If it does not provide converged springback results, the user may select the multi-step implicit method for the springback analysis.

The coarsen control can be activated to combine the refined mesh prior to springback analysis. The function allows the eta/DYNAFORM solver to combine neighbouring elements that have an difference of normal angle of less than a given angle (default is 8°). Coarsening blank mesh can reduce computational noise, and therefore helps the springback simulation to converge.

### CONSTRAINT

Enable the user to select appropriate constraint points to prevent rigid body motion during springback analysis.

### IMPLICIT PARAMETERS

Define parameters for springback analysis. The dialog box is shown in Figure 8.1.11. For detailed information about each parameter, refer to the LS-DYNA User's Manual.

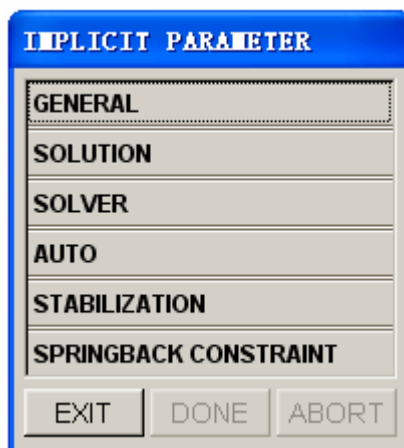


Figure 8.1.11 The Implicit Parameters dialog box

#### 8.1.3.1 THE QUICK SETUP/SPRINGBACK PROCEDURE

Following is the procedure for setting up a springback analysis using QS/SPRINGBACK:

1. Read in the blank results.  
The blank results are obtained from a draw simulation. They are output as a DYNAIN file after each draw simulation. The user may read in the DYNAIN file by selecting BLANK from the QS/SPRINGBACK tool GUI and then selecting IMPORT MESH from the DEFINE BLANK dialog box.
2. Trim the blank.  
If no trim operation is involved, the user may skip this step and continue with Step 3. Refer to Section 10.8.5 for more information about blank trimming.
3. Define blank parameters.  
The user needs to define the initial blank material and thickness. The blank parameters are required for the solver to perform initialization prior to springback analysis. These blank parameters should be similar to the blank parameters used in the draw simulation.. After the initialization, the thickness, stress and strain stored in the DYNAIN file will be used as the initial blank parameters.

4. Select type of implicit method and coarsen options.

5. Define implicit parameters.

There are no appropriate default implicit parameters which are good for all springback applications. The user needs to adjust implicit parameters accordingly. Refer to the LS-DYNA User's Manual for a detailed explanation of all the implicit parameters following the listed cards. NOTE: control cards list different solver versions.

6. Control rigid body motion.

The user needs to define appropriate constraints on the blank to eliminate the six rigid body motions. This is done by using the SPRINGBACK CONSTRAINT option in the IMPLICIT PARAMETER dialog box (Figure 8.1.11). eta/DYNAFORM will prompt the user to select the constrained nodes.

For most applications, 3 constraint nodes are recommended for springback analysis. The chosen 3 constraint nodes should be well separated from each other and away from the edges and flexible areas in the part. The first constraint node will lock the three translation degrees of freedom and define the reference point in the springback model. The springback displacement is zero at the first constraint node. The second constraint node will lock the Y- and Z- translation, while the third constraint node eliminates the translation in Z-direction. For detailed information about springback analysis, contact eta/DYNAFORM's Technical Support at [support@eta.com](mailto:support@eta.com).

## 8.2 AUTO SETUP

Auto setup is a newly added module in eta/DYNAFORM Version 5.5 to make setup more conveniently. This module includes most basic functions in all traditionally setups and the hydro-forming module. In addition, simple multi-process simulations and other new features such as user defined stamping direction and welded blank simulation are also added to Auto Setup. In the new Auto Setup module, users only need to define tools and direct tool closing to define calculation time for each process. It greatly saves users' simulation setup steps and reduces risk of making mistakes.

The new Auto Setup interface consists of one menu bar and five main setup pages, which are General, Blank, Tools, Process and Control.

### 8.2.1 NEW SIMULATION

The user needs to define a new simulation before adding to Auto Setup. Click SETUP/AUTOSETUP menu, if there is no existing Auto Setup information in user's database, the program will automatically add new simulation and guide user to setup. If there is defined Auto Setup information in the database, the program will automatically enter the defined Auto Setup to edit.

**NOTE:** *The user is allowed to set only one Auto Setup project in one eta/DYNAFORM database.*

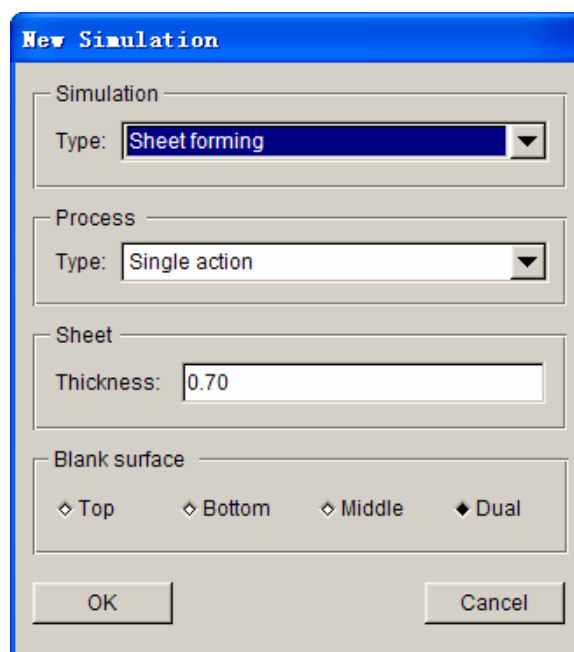


Figure 8.2.1 New Auto Setup Setting

#### 8.2.1.1 SIMULATION TYPE

There are two simulation types in the new Auto Setup. One is sheet forming, which include traditional

normal sheet forming and blank hydro forming. The other one is tubular hydro forming.

### 8.2.1.2 PROCESS

The new Auto Setup includes 6 ordinary process templates and an empty template as shown in Figure 8.2.2.

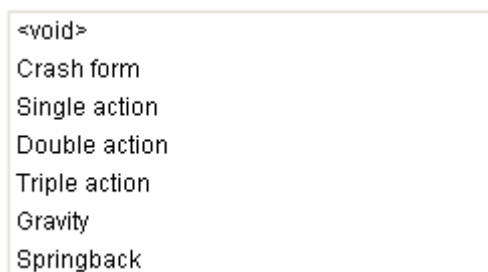


Figure 8.2.2 Process

**<VOID>**: program does not provide any default setup. Users may define their own set up according to their requirement.

**CRASH FORM**: draw forming without closing ring.

**SINGLE ACTION**: single action forming simulation.

**DOUBLE ACTION**: double action forming simulation.

**TRIPLE ACTION**: triple action forming simulation.

**GRAVITY**: gravity forming simulation

**SPRINGBACK**: springback simulation.

### 8.2.1.3 BLANK THICKNESS

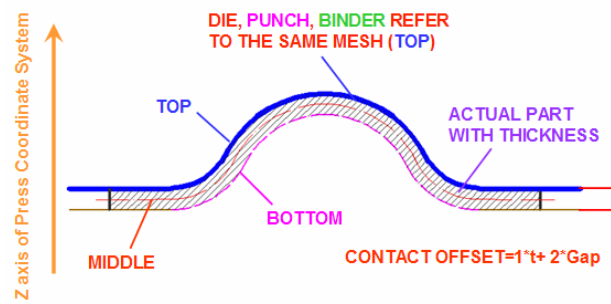
At the beginning of a new setup, the program will prompt user to enter a default blank thickness. If the default blank thickness is defined, this value will be automatically set to the defined blank. The user may go to blank define page to modify a certain blank thickness. Default blank thickness is 1.0 (mm).

### 8.2.1.4 BLANK SURFACE

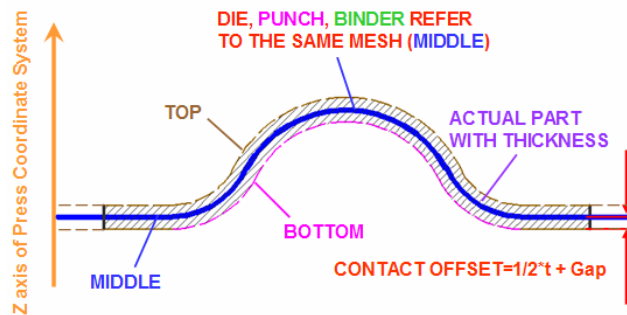
In real simulation process, the user usually generates die and punch surface automatically from the original part surface. There is only one part surface. However, in practical operating process, die and punch have different surfaces. It is required to define side of the part surface which is used to generate die face: top, middle or bottom. The program will automatically set up the contact OFFSET value on the other side of the surface.

If physical offset die and punch surface are used, the user should select the fourth option DUAL. Refer to the Figure 8.2.3 for more explanation.

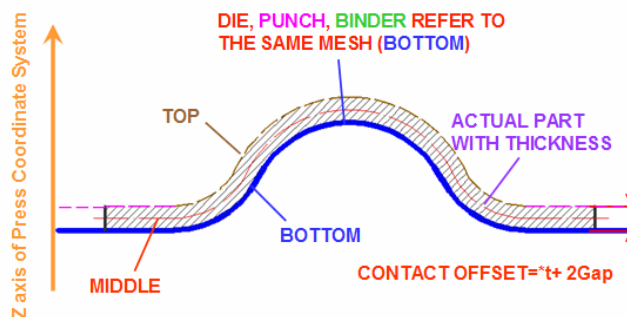




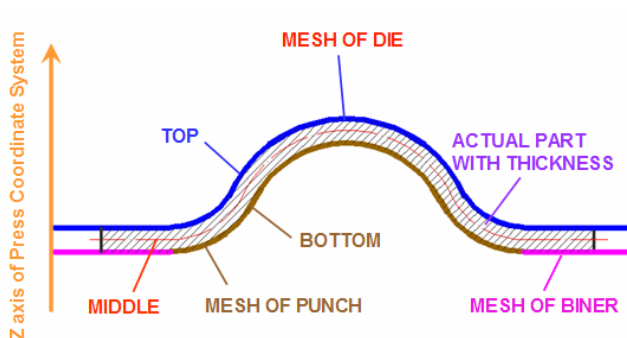
a) Tools refer to the TOP surface of part



b) Tools refer to the MIDDLE surface of part



c) Tools refer to the BOTTOM surface of part

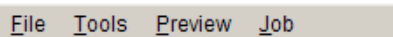


d) Tools refer to the DUAL surface of part

Figure 8.2.3 Blank Surface

## 8.2.2 AUTO SETUP MENU

In the Auto Setup interface, the menu bar is provided to enable the user to quickly inquire ordinary functions. The menu bar includes functions such as save, new, tool, draw beam define, dynamic display and job submission.



### 8.2.4 Auto Setup Menu Bar

#### 8.2.2.1 FILE

File menu includes New, Save and Exit Auto Setup.

**NEW** — Create a new setup. The program will clear all existing setup. The program will display a dialog box to confirm the action.

**SAVE** — Save current setup. Program will save the current data to the database file.

**CLEAR** — Delete all the setting in AUTO SETUP.

**EXIT** — Exit Auto Setup dialog box and return to eta/DYNAFORM main interface.

#### 8.2.2.2 TOOLS

The Tool Menu includes Displaying, Homing, Positioning, Drawbead.

**DISPLAYING** — Control tool display. The user may conveniently control tool display on and off as shown in Figure 8.2.5.

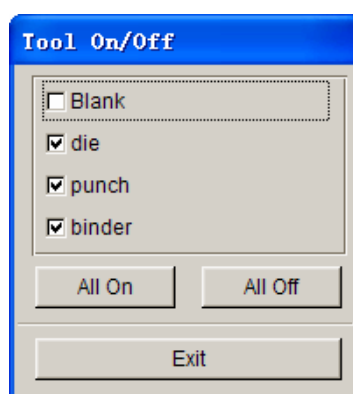


Figure 8.2.5 Tool On/Off

**HOME** — This function is used for adjusting the relative position of the die during closing stage. If the user defines certain tool to its HOME position as the calculation time of this process, he/she needs this function to adjust relative position between each tool. The program

will display the Tool Moving dialog box as shown in Figure 8.2.6.

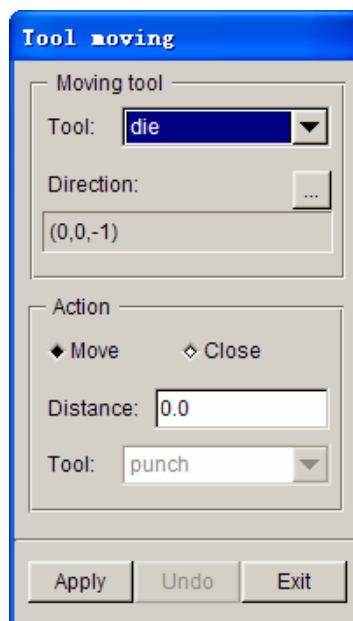



Figure 8.2.6 Tool Moving dialog box

**Moving tool:** Select tool to adjust initial position.

**Direction:** Select tool moving direction. The default moving direction of each tool is its working direction. The user may also define moving direction according to the requirement. Click  button to open the DEFINE DIRECTION dialog box as illustrated in Figure 8.2.7.

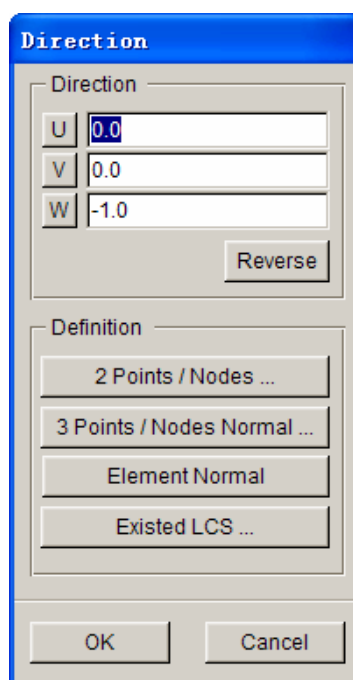





Figure 8.2.7 Define Direction dialog box

 — Define X coordinate of direction vector. The user may click this button to set the default direction to be +X axis direction. If click the button again, the direction is changed in -X direction. The user may directly enter the known coordinate in the input box.

 — Define Y coordinate of direction vector. After the user clicked this button, the default direction is +Y direction. If the user double click the button, the direction will change in -Y direction. The user may directly enter the known coordinate in the input box.

 — Define Z coordinate of direction vector. After clicking this button, the default direction is +Z direction. If the user double click this button, the direction will change in -Z direction. The user may directly enter the known coordinate in the input box.

**Reverse** — Reverse defined coordinates' direction.

**Two Points/Nodes** — Define vector direction by selecting two points or two nodes on the screen. The first point is the starting point of vector and the second one is the end point.

**Three Points/Nodes Normal** — Define vector direction by plane normal that is set by three selected points or nodes. After clicking this function, system will prompt the user to select three points or nodes.

**Element Normal** — Select an element to define a vector by the element's normal. After clicking this button, the user needs to select an element on the screen. After selecting the element, the vector direction is highlighted at the center of the element.

**Existed LCS.** — This function allows the user to select a defined LCS X, Y, Z axis to be the direction vector. If there is no defined LCS in the database, the button turns gray. The user may use menu UTILICITIES/COORDINATE SYSTEM to define local coordinate system. After clicking this button, the program displays the select coordinate system dialog box as shown in Figure 8.2.8.

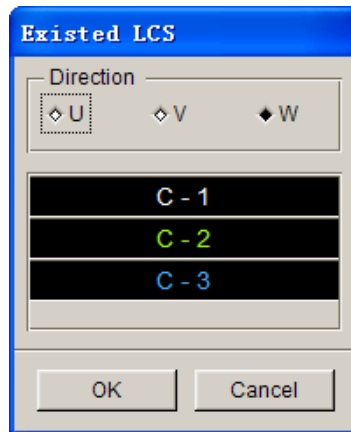


Figure 8.2.8 Select Existing LCS

The user may select any coordinate system by mouse clicking it, following by assigning the vector direction by selecting U, V or W toggle.

**Action:** Select tool-moving action. There are two moving actions. One is free move. The other is automatic move by assigning two closing tools.

**Distance:** If the user selects free move tool, the DISTANCE is the tool travel distance after the APPLY button is clicked. The user may repeat clicking the APPLY button to move the defined tool. The user may click the UNDO button to cancel the previous move. If the user selects automatic tool moving, the DISTANCE is the distance between the two tools in the moving direction after the auto move. For example, if the user adopts offset to create die and punch directly, a blank thickness is set as this Figure to auto adjust the Home position of die.

**Tool:** This selection allows user to define a reference tool. This tool is mainly used to reference another tool moving while auto-moving the tool.

**POSITIONING** — This function is used to adjust the relative position of die during die travel. This function includes auto positioning function and manual positioning. For detailed description of positioning function of tool, refer to Section 8.2.5.5.

**DRAWBEADS** — This function allows the user to set up drawbeads. Detailed descriptions about drawbeads are provided in Section 10.4.

### 8.2.2.3 PREVIEW

This function is used to check the tool and process setup condition. The ANIMATION function allows the user to preview each tool's moving condition. The SUMMARY function allows the user to review the whole setup information.

**ANIMATION** — Animate the moving condition of tool. After selecting this button, the

system displays the animate dialog box as shown in Figure 8.2.9. Detailed descriptions of the animation control are provided in Section 10.11.

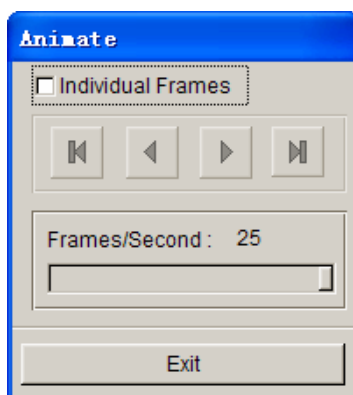


Figure 8.2.9 Animate dialog box

**SUMMARY** — This function lists basic information of the user’s current setup, which includes stamping direction, tool, blank, process name and contact, etc. A typical Summary output is shown in Figure 8.2.10.

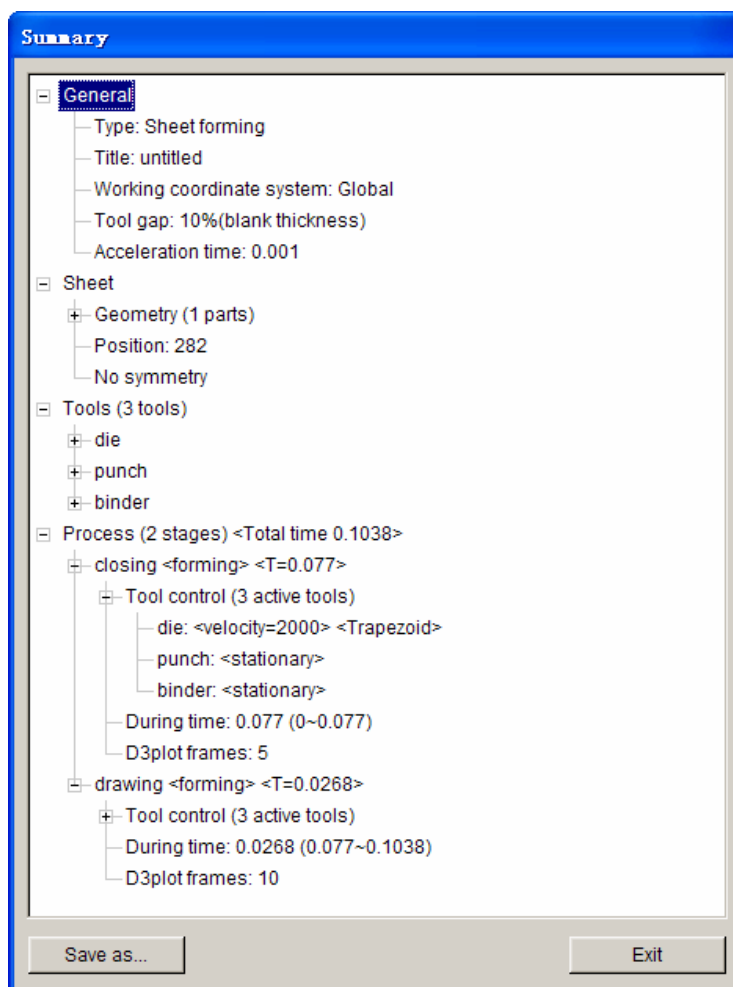


Figure 8.2.10 Summary output

**8.2.2.4 JOB**

This menu includes LS-DYNA INPUT FILE, FULL RUN DYNA and JOB SUBMITTER functions.

**LS-DYNA INPUT FILE** — This function enables the user to output the current job to a DYNA input file. A file dialog box is displayed for the user to specify the path and directory to store the output file.

**FULL RUN DYNA** — This function enables the user to run the current job immediately. The program will launch the LS-DYNA solver to run the current job.

**NOTE:** *The user should define the appropriate directory for the LS-DYNA solver in the TOOLS/ANALYSIS SETUP menu.*

**JOB SUBMITTER** — This function allows the user to submit the current DYNA job to the JOB SUBMITTER. Then, he/she may manage the DYNA jobs through the JOB SUBMITTER. Submit job dialog box is shown in Figure 8.2.11.

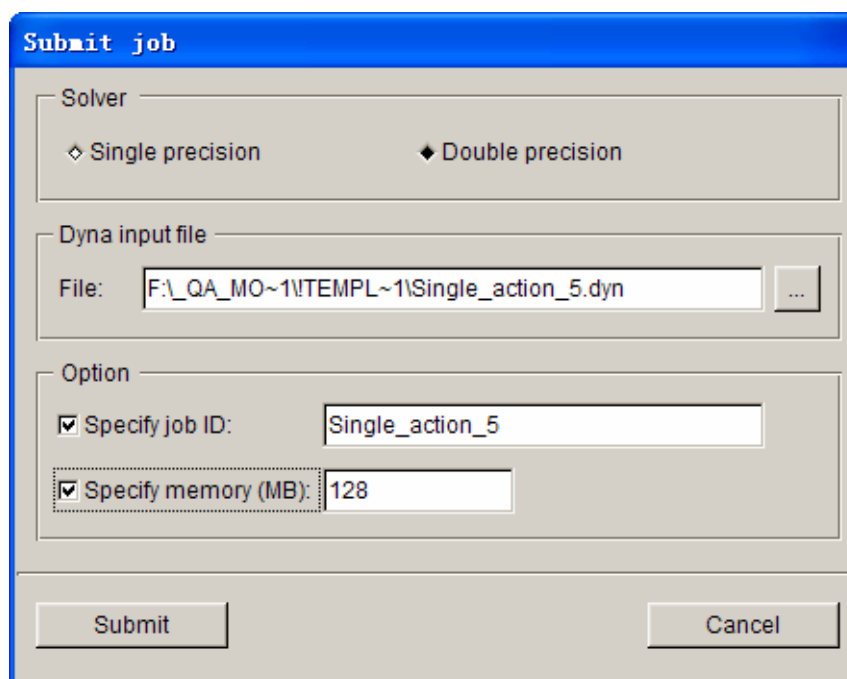



Figure 8.2.11 Submit Job dialog box

**Solver:** Enables the user to select the Single or Double Precision solver to run the current job. The user may also change the solver type in the JOB SUBMITTER.

**Dyna input file:** Define the directory and file name of the output DYNA file. The

default output directory is the current working directory. The user may click  button behind to change the output directory.

**Specify job ID:** The user may select whether to specify an ID for the current job. If there are more than one simulation result under the same directory, the result will not overwrite the previous result files.

**Specify memory (MB):** Specifies memory size for current job. The user may specify different sizes according to the model size.

**Submit:** Allows the user to submit the current job. The program will display the Job Submit interface, and add the current job to the job list.

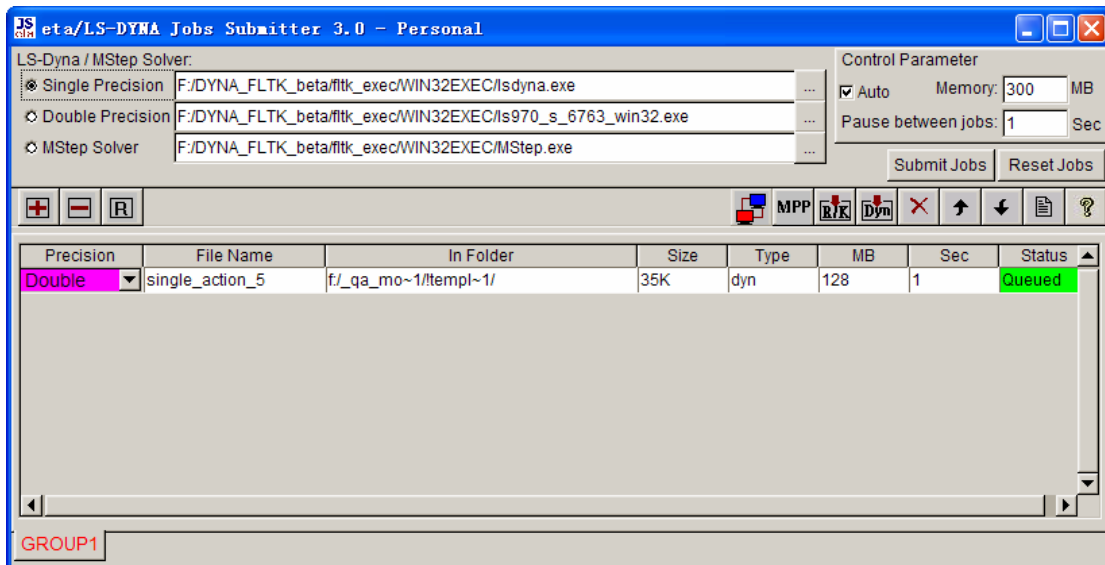


Figure 8.2.12 Job Submitter

**Cancel:** Exits Submit Job interface and return to Auto Setup interface.

### 8.2.3 GENERAL

In GENERAL definition page, the user may define basic information for the current setup, including: title, working coordinate system, tool acceleration time and comment. Figure 8.2.13 illustrates a typical general setup GUI window.



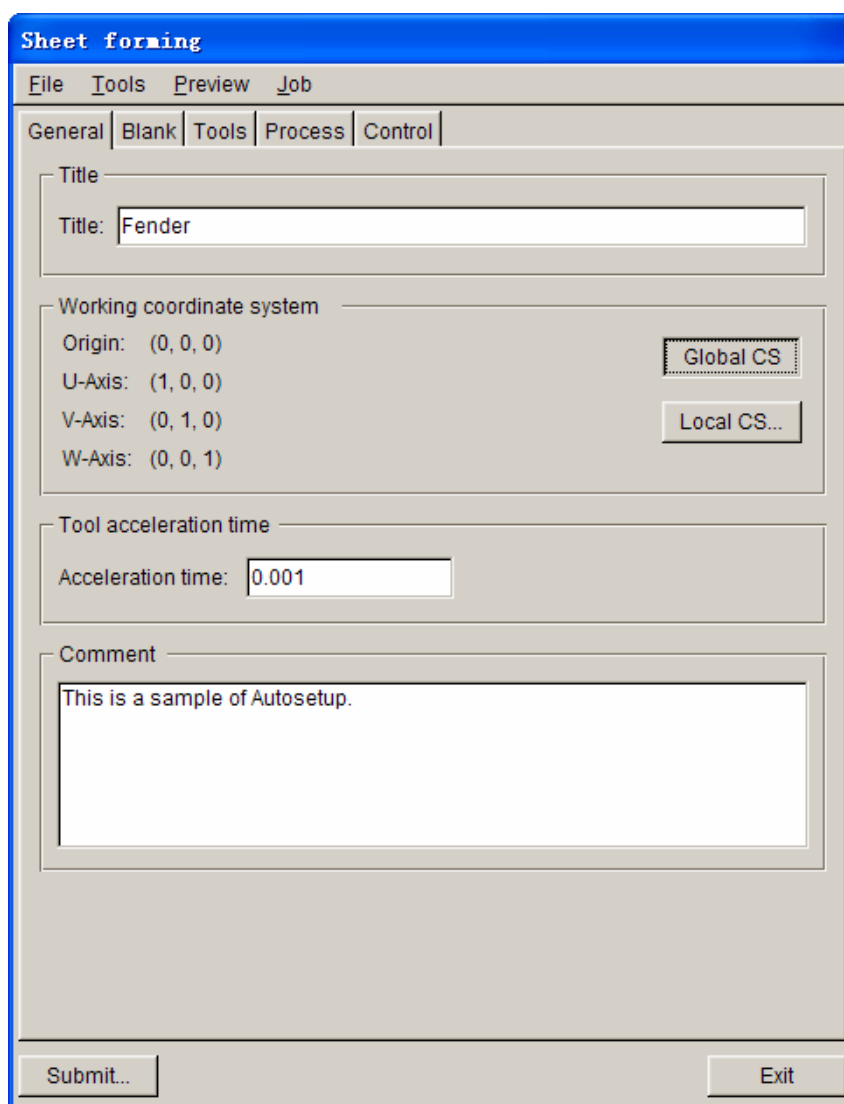


Figure 8.2.13 General definition page

### 8.2.3.1 TITLE

This function allows the user to enter a title name for the current setup. The user may use any meaningful title to describe the setup.

### 8.2.3.2 WORKING COORDINATE SYSTEM

This function allows user to set up the working coordinate system. Earlier eta/DYNAFORM versions do not support arbitrary direction as the stamping direction. It is not necessary for the user to transform the forming result to the actual position of the part in the automobile to carry out crash analysis. The user may define a LCS, and use local W direction as the stamping direction. Default coordinate system is global Z direction. If the user needs to define local coordinate system, click the Local CS button to open the WCS dialog box as shown in Figure 8.2.14.

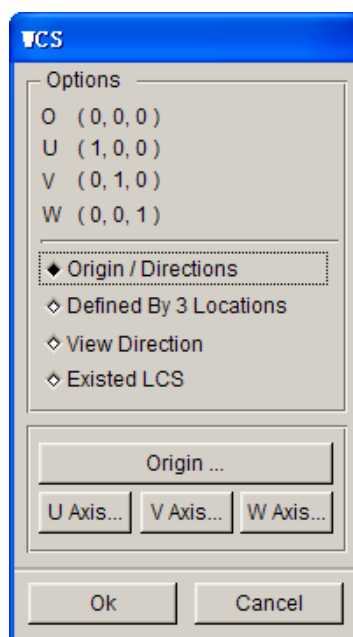


Figure 8.2.14 WCS dialog box

**ORIGIN/DIRECTIONS** — Define LCS by defining the origin and the direction of coordinate axes. The user may only define W direction to be parallel to local X or Y axis and global X or Y axis.

**Origin...:** This button allows the user to define local origin. After clicking this button, the INPUT dialog box is displayed. The user may enter point coordinates to define origin or select a node or point by mouse clicking.

**U Axis...:** This function allows the user to define local U direction. After clicking the button, the DEFINE DIRECTION dialog box as shown in Figure 8.2.15 is displayed.

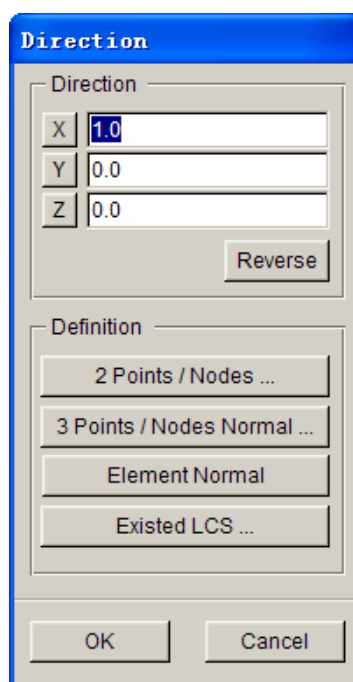


Figure 8.2.15 Direction dialog box

**DIRECTION**

Allow the user to define a vector to specify the direction. The starting point of the vector is at the current coordinate system's origin. The user needs to define the coordinates of the end point of the vector.

- Define the X coordinate of the vector's end point .
- Define the Y coordinate of the vector's end point.
- Define the Z coordinate of the vector's end point.

**REVERSE**

Allow the user to reverse the current defined direction.

**2 POINTS/NODES**

Allow the user to select two points or nodes to define a direction.

**3 POINT / NODES NORMAL**

Allow the user to select three points or nodes to define a direction. This direction is normal to the plane defined by the three points, according to the right hand rule.

**ELEMENT NORMAL**

Allow the user to use the normal direction of the selected element as the direction.

**EXISTED LCS.**

Allow the user to select a coordinate from defined LCS as the required direction. After clicking this button, the LCS dialog box as shown in Figure 8.2.16 is displayed. The user may select any LCS and it's direction.

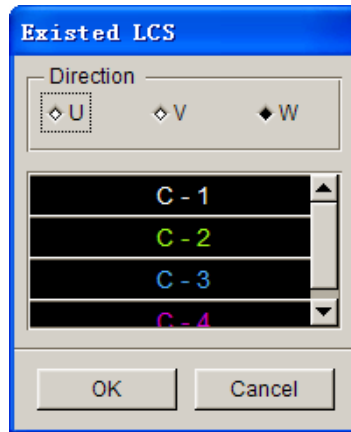


Figure 8.2.16 Select LCS dialog box

**U Axis...:** This function allows the user to define local U axis direction. After clicking this button, the DEFINE DIRECTION dialog box is displayed. The user may select any of the LCS in the list.

**V Axis...:** This function allows the user to define local V axis direction. After clicking this button, the DEFINE DIRECTION dialog box. The operation is the same as in U axis.

**W Axis...:** This function allows user to define local W axis direction. After clicking this button, the DEFINE DIRECTION dialog box is displayed. The operation is the same as U and V axes.

*Note: The user may define U axis and V axis to define LCS. W direction is calculated according to the right hand rule.*

$$W=U \times V$$

*The V axis is adjusted again from the W and U axis according to the right hand rule.*

$$V=W \times U$$

*The user may only define W axis direction to define the LSC. The U, V direction is defined automatically. This option is useful when the local U and V direction are not important.*

**DEFINED BY 3 LOCATIONS** — Specifies a normal direction by defining three points.

This direction is used as the working coordinate system. The interface is shown as in Figure 8.2.17:

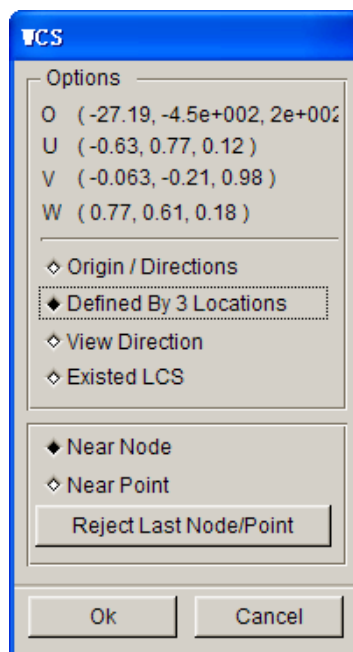


Figure 8.2.17 WCS

**Near Node:** Defines WCS by selecting three nodes.

**Near Point:** Defines WCS by selecting three geometry points.

**Reject Last Node/Point:** Rejects last selected node/point.

**VIEW DIRECTION** — This function allows the user to use current view direction as WCS. Local W axis points to the user from the screen.

**EXISTED LCS** — This function allows the user to select existing LCS and use the local W axis as WCS as shown in Figure 8.2.18.

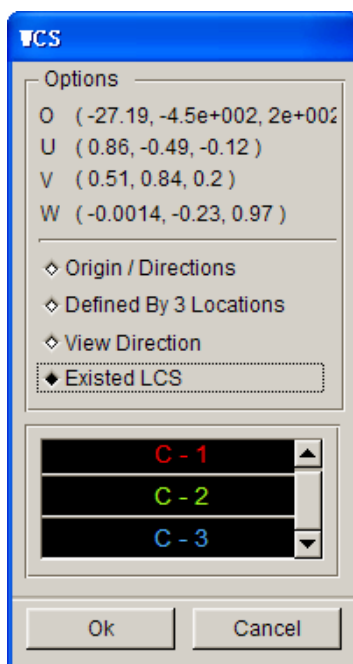


Figure 8.2.18 Define WCS by Selecting LCS

**8.2.3.3 TOOL ACCELERATION TIME**

This function allows the user to specify the tool acceleration time. When defining travel curve in LS-DYNA, the speed ramps up from 0 to the peak value. It is better to define the ramp time to guarantee the stability of calculation. Default ramp time is 0.001 second as shown in Figure 8.2.19.

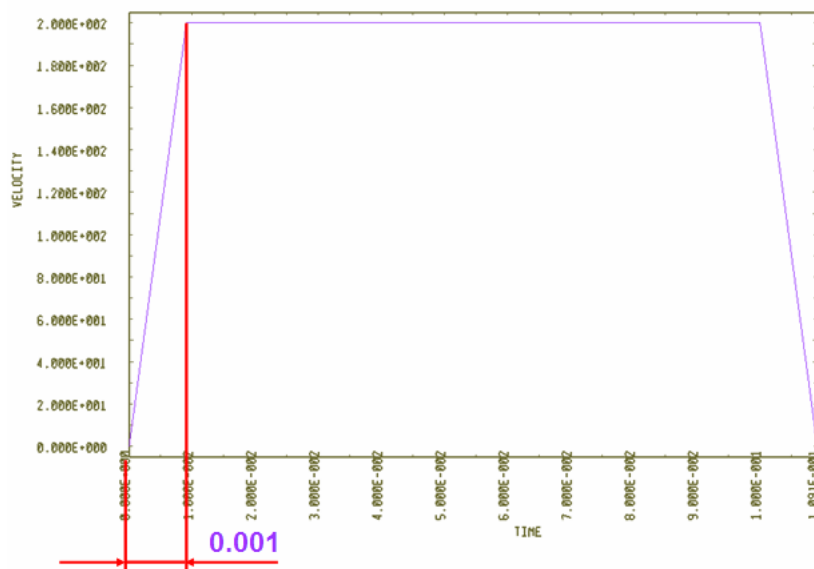


Figure 8.2.19 Acceleration Time Setup

**8.2.3.4 COMMENT**

This function enables the use to write comments about the current setup.

## 8.2.4 BLANK

In the BLANK definition page, the user may define part, material, thickness, weld joint between multiple blanks and blank symmetry condition. Its interface for blank definition is shown in Figure 8.2.20.

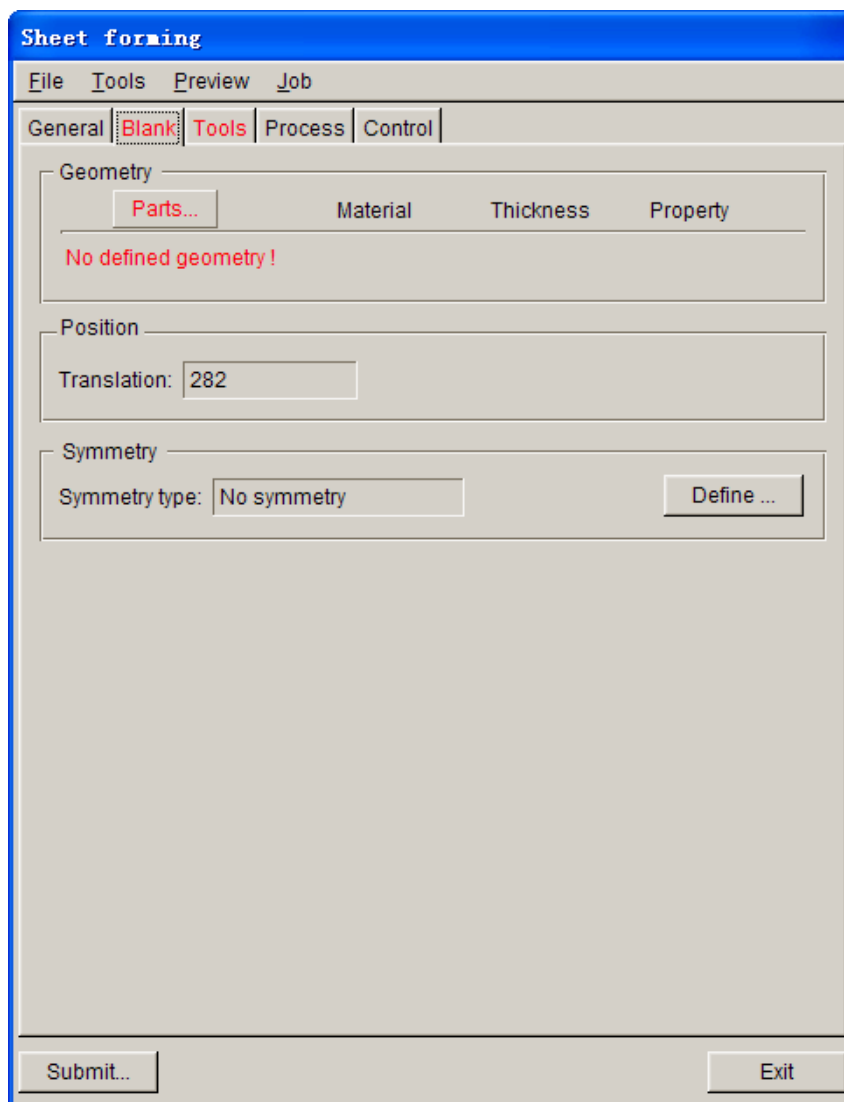


Figure 8.2.20 Blank definition page

### 8.2.4.1 GEOMETRY

#### PARTS

Allow the user to define blank by selecting part or import mesh file directly. After clicking the **Parts...** button as illustrated in Figure 8.2.21, the DEFINE GEOMETRY dialog box, as shown in Figure 8.2.22, is displayed.

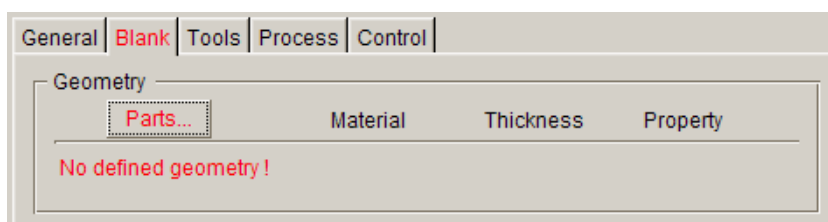


Figure 8.2.21 Blank Part

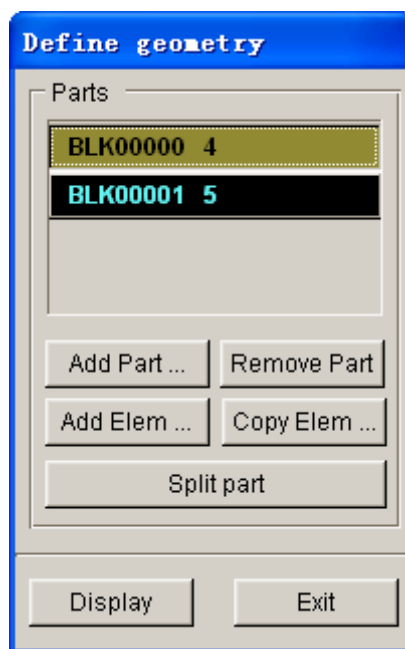


Figure 8.2.22 Define Geometry dialog box

**ADD PART...:** This function allows user to define blank by selecting one or more parts from the part list. Selected parts are highlighted in parts list and on the screen.

**REMOVE PART:** This function allows the user to remove selected parts from the part list.

**ADD ELEM...:** This function allows the user to add selected elements to selected part.

**COPY ELEM...:** This function allows the user to copy or offset new elements from existing elements. New elements will be added to the part, which is defined as blank.

**SPLIT PART:** This function allows user to split the blank into several blanks, and then the blanks can be defined different materials and properties. So the user can easily generate the blank of weld forming simulation.



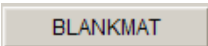
**DISPLAY:** Allow the user to display all the parts defined as the blank. The user may easily check and repair the mesh without displaying other parts on the screen.

**EXIT:** Allow the user to dismiss the DEFINE GEOMETRY dialog box.

After the user defines blank parts, all defined parts are displayed in the list under Part. A default material and property is assigned for the part. The user may change the default material, thickness and property for each part.

## MATERIAL

The Auto Setup defines a default type 36 material for blank part. The user may click

 under material to modify or create a material as shown in Figure 8.2.23.

Material definition interface is shown in Figure 8.2.24.



Figure 8.2.23 Define material

**NOTE:** If the Part has no defined material, the material bar will display None in red. The program will prompt the user to define material for this part. In Auto Setup, the user must define the parameters labeled in red. After defining the material, the material name is displayed on the button.

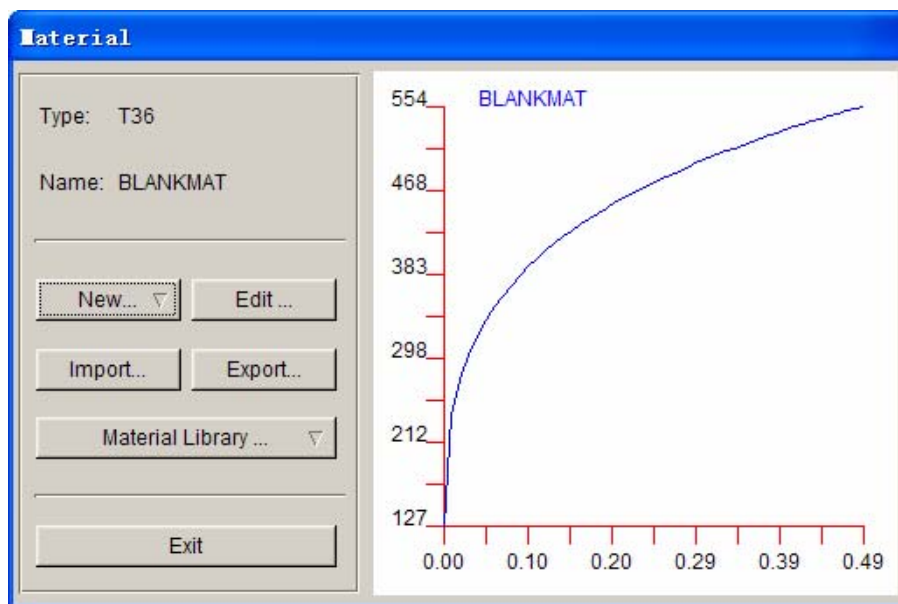


Figure 8.2.24 Material Define dialog box

**TYPE:** Displays current user defined material type. eta/DYNAFORM supports

several sheet forming material types of LS-DYNA, such as: 1, 18,24, 36, 37, 39 and 64.

**NAME:** Material name. Displays the defined material name.

**NEW...:** Allows the user to setup a new material. After clicking this button, the program will display the Type list. The user may select one of the types from the list as shown in Figure 8.2.25. The MATERIAL dialog box as shown in Figure 8.2.26 is displayed.

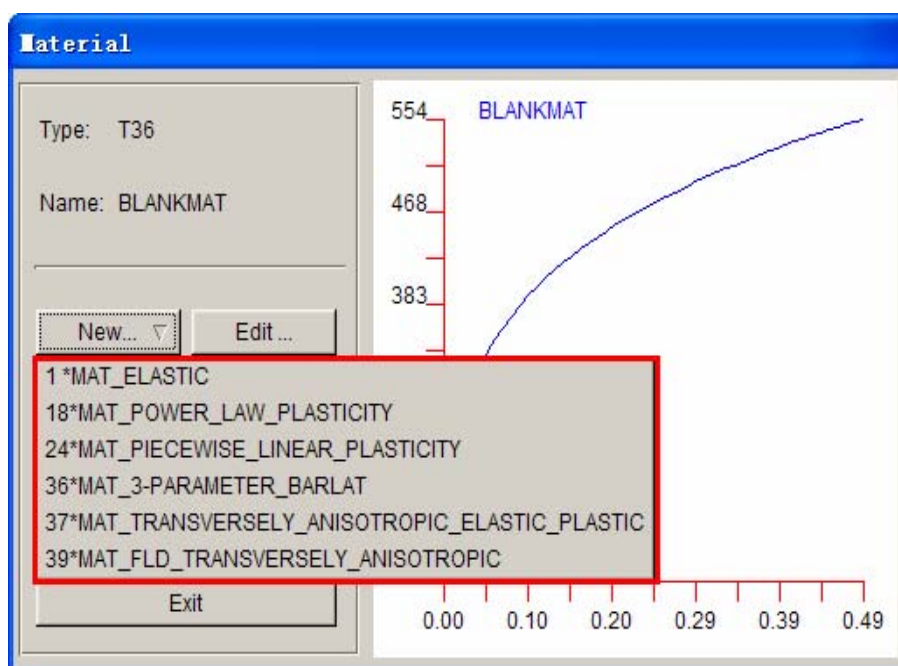



Figure 8.2.25 Select material type

The image shows a 'Material' dialog box with the following fields and values:

Field	Value
Type	36*MAT_3-PARAMETER_BARLAT
Material Name	BLANKMAT
Mass Density	7.830000e-009
Young's Modulus	2.070000e+005
Poisson's Ratio	2.800000e-001
Hardening Rule (Expon)	2.000000
Material Param P1 (K)	6.480000e+002
Material Param P2 (N)	2.200000e-001
Exponent Face (M)	6.000000
Lankford Param R00	1.870000
Lankford Param R45	1.270000
Lankford Param R90	2.170000
Strain/stress curve	<none>
Forming limit curve (FLC)	<none>

Buttons at the bottom: Advanced..., Default, OK, Cancel

Figure 8.2.26 Material Define

**Forming Limit Curve (FLC):** This parameter is only necessary for material type 39. At this time, <NONE> is displayed behind FLC curve to indicate there is no FLC curves defined. If the user needs to define FLC curve for post processor, click  button to create a FLC curve. When the user enters key word files, the program automatically defines FLC information, including FLC curve, FLD0 (true and nominal/engineering) which are output to index file. The post processor will load the FLC information though the index file written by eta/DYNAFORM.

**NOTE:** The FLC curve defined in Auto Setup is represented in engineering strain.

After entering FLC curve edit dialog box, the user may enter coordinates of the data points directly in INPUT page to define the FLC curve. Else, the user may switch to FORMULA page to define FLC curve using KEELER's empirical formula. After the user enters appropriate **n** and **t** value, click the APPLY button to generate the FLC. The FLC curve is displayed in the show curve interface as shown in Figure 8.2.27. Detailed description of curve manager operation is provided in Section 8.2.6.5.

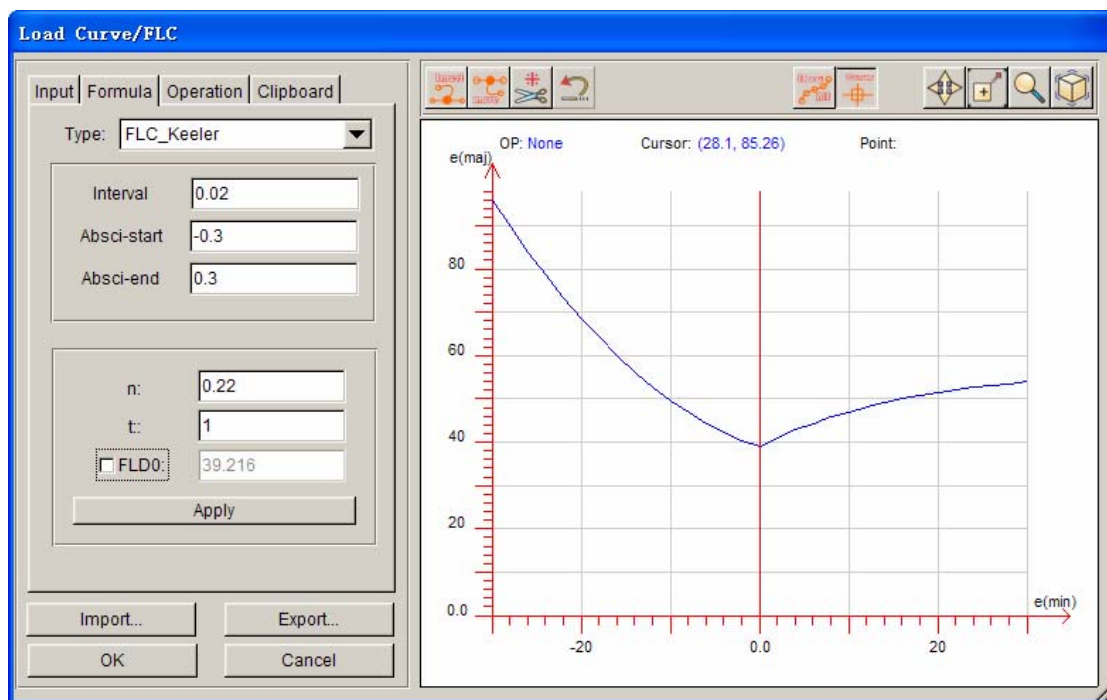
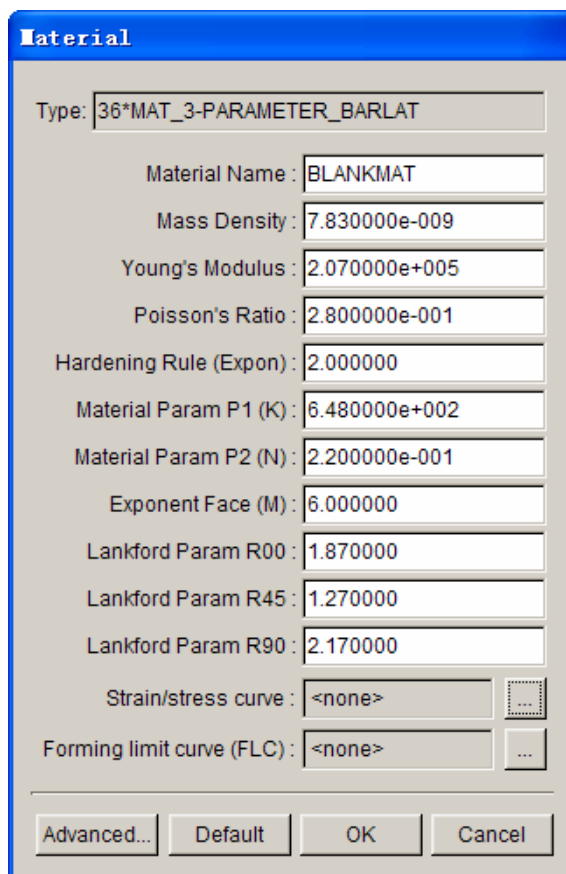


Figure 8.2.27 FLC Curve

**EDIT...:** Allow the user to edit the current material. After clicking this button, the MATERIAL dialog box as shown in Figure 8.2.28 is displayed.



The image shows a 'Material' dialog box with a blue title bar. The 'Type' field is set to '36\*MAT\_3-PARAMETER\_BARLAT'. Below this are several input fields for material properties: 'Material Name' (BLANKMAT), 'Mass Density' (7.830000e-009), 'Young's Modulus' (2.070000e+005), 'Poisson's Ratio' (2.800000e-001), 'Hardening Rule (Expon)' (2.000000), 'Material Param P1 (K)' (6.480000e+002), 'Material Param P2 (N)' (2.200000e-001), 'Exponent Face (M)' (6.000000), 'Lankford Param R00' (1.870000), 'Lankford Param R45' (1.270000), and 'Lankford Param R90' (2.170000). There are also two dropdown menus for 'Strain/stress curve' and 'Forming limit curve (FLC)', both currently set to '<none>'. At the bottom, there are four buttons: 'Advanced...', 'Default', 'OK', and 'Cancel'.

Field	Value
Type	36*MAT_3-PARAMETER_BARLAT
Material Name	BLANKMAT
Mass Density	7.830000e-009
Young's Modulus	2.070000e+005
Poisson's Ratio	2.800000e-001
Hardening Rule (Expon)	2.000000
Material Param P1 (K)	6.480000e+002
Material Param P2 (N)	2.200000e-001
Exponent Face (M)	6.000000
Lankford Param R00	1.870000
Lankford Param R45	1.270000
Lankford Param R90	2.170000
Strain/stress curve	<none>
Forming limit curve (FLC)	<none>

Figure 8.2.28 Material Edit dialog box

**NOTE:** Material edit dialog box and material define dialog box are basically similar. The user may modify any parameter in the dialog box, except the material type. If the user needs to change the material type, he/she must redefine the material properties.

**IMPORT:** Allow the user to import a material file. The extension of the file name is “.mat”. After clicking this button, the program displays the import material window as shown in Figure 8.2.29.

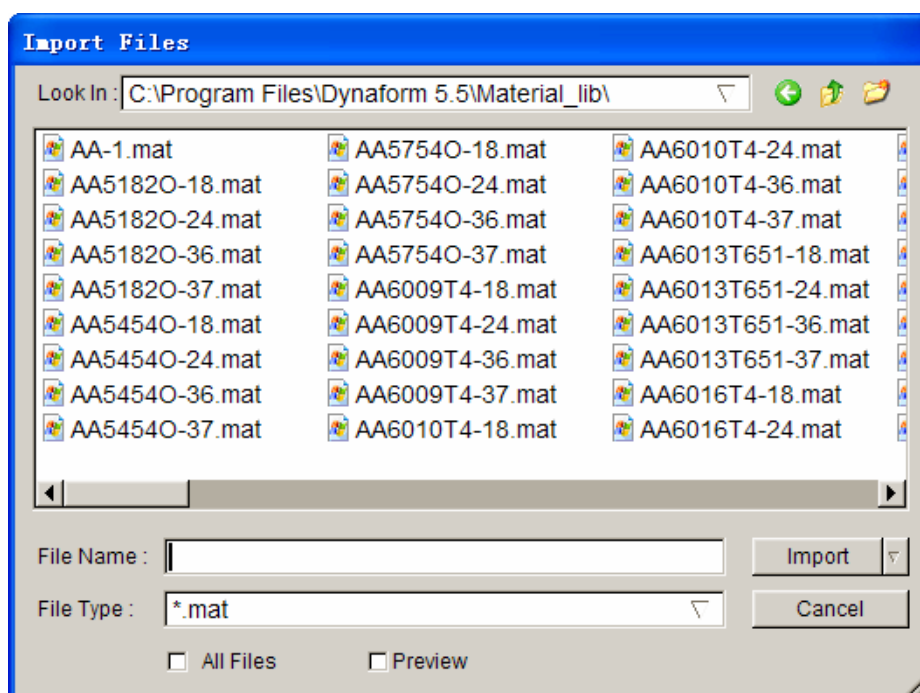


Figure 8.2.29 Input Material window

**PREVIEW:** The user may preview stress strain curve of selected material. See Figure 8.2.30.

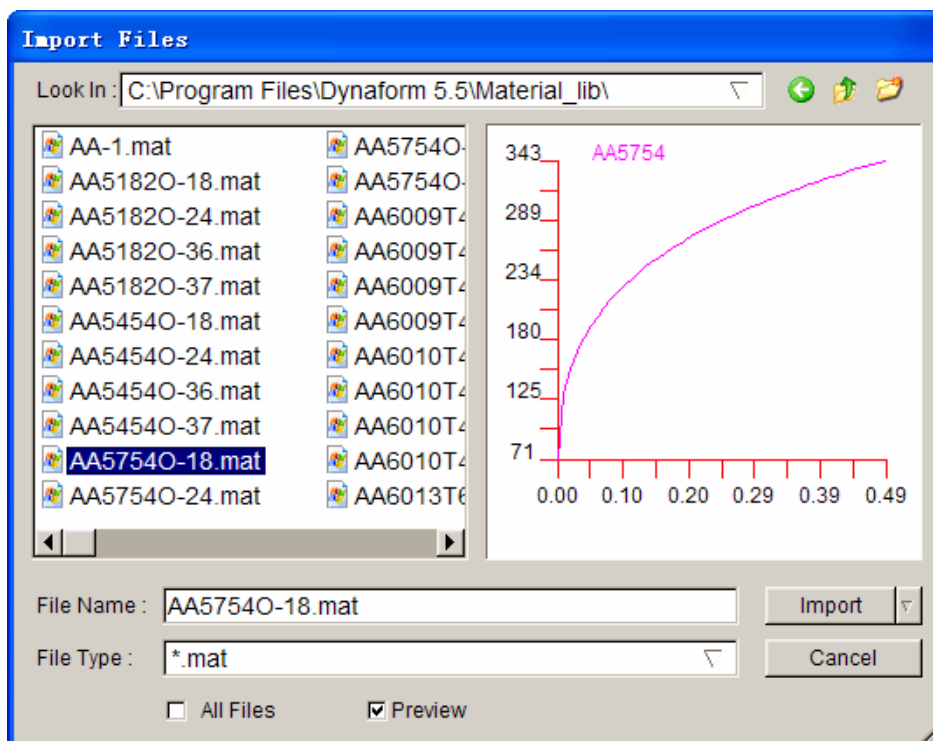


Figure 8.2.30 Material Stress-Strain Curve Preview

If the user defines a material for the blank, the preview function will activate the comparison option. This option allows the user to compare the stress strain curve of the defined material and the selected material from the library. See Figure 8.2.31 for a sample comparison.

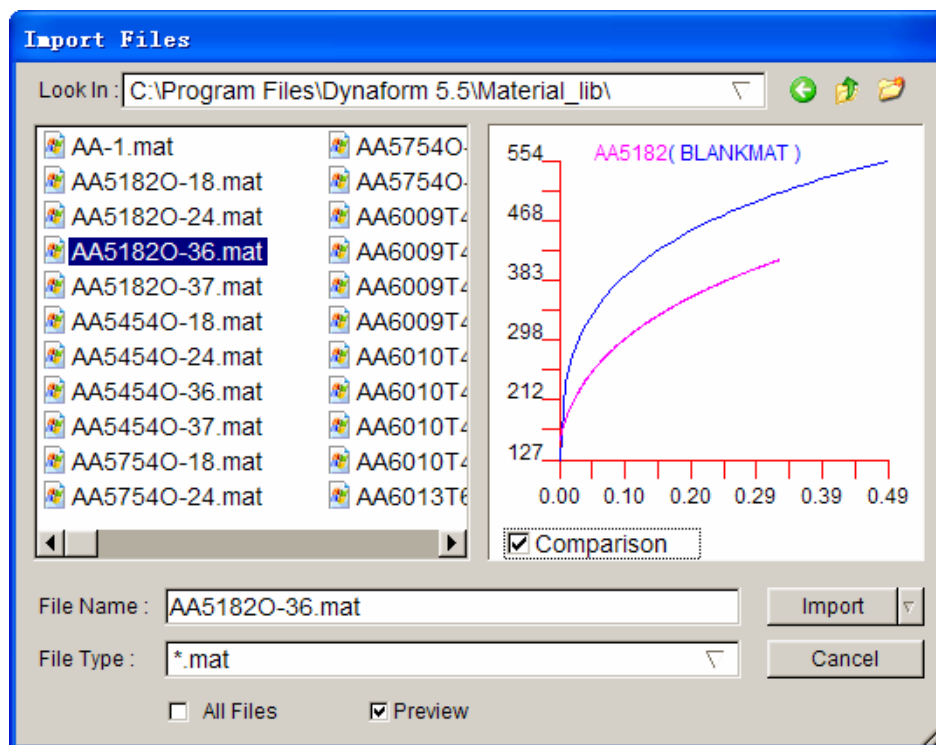


Figure 8.2.31 Material Stress Stain Curve Comparison

The magenta curve is selected reference material curve. The blue curve is the user defined material curve. The name of current selected material and defined material are both displayed on the top of preview window. Name listed in the parentheses is the name of defined material.

After selecting a file, the user may click IMPORT button to import selected material and assign it to selected blank part.

**EXPORT:** Allow the user to export current material to a file. File is saved with an extension “.mat”. See Figure 8.2.32 for the export GUI window.

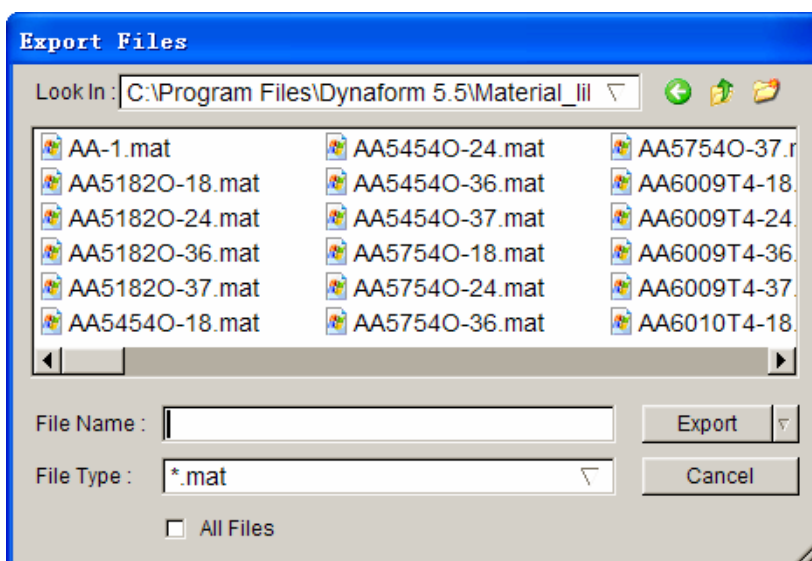


Figure 8.2.32 Export Material to a File

**MATERIAL LIBRARY:** This function enables the user to select a material from the native material library in eta/DYNAFORM. The material standard list will pop up after clicked the Material Library... button as shown in Figure 8.2.33 shown. The user can select a material standard from the list. There are four type of standard in eta/DYNAFORM. The available standards are U.S., JAPAN, CHINA and EUROPE.



Figure 8.2.33 Material Library Standard

After selecting a material library standard, the program will display the material library GUI as shown in Figure 8.2.34. Then, the user may select one of materials from library for the blank.



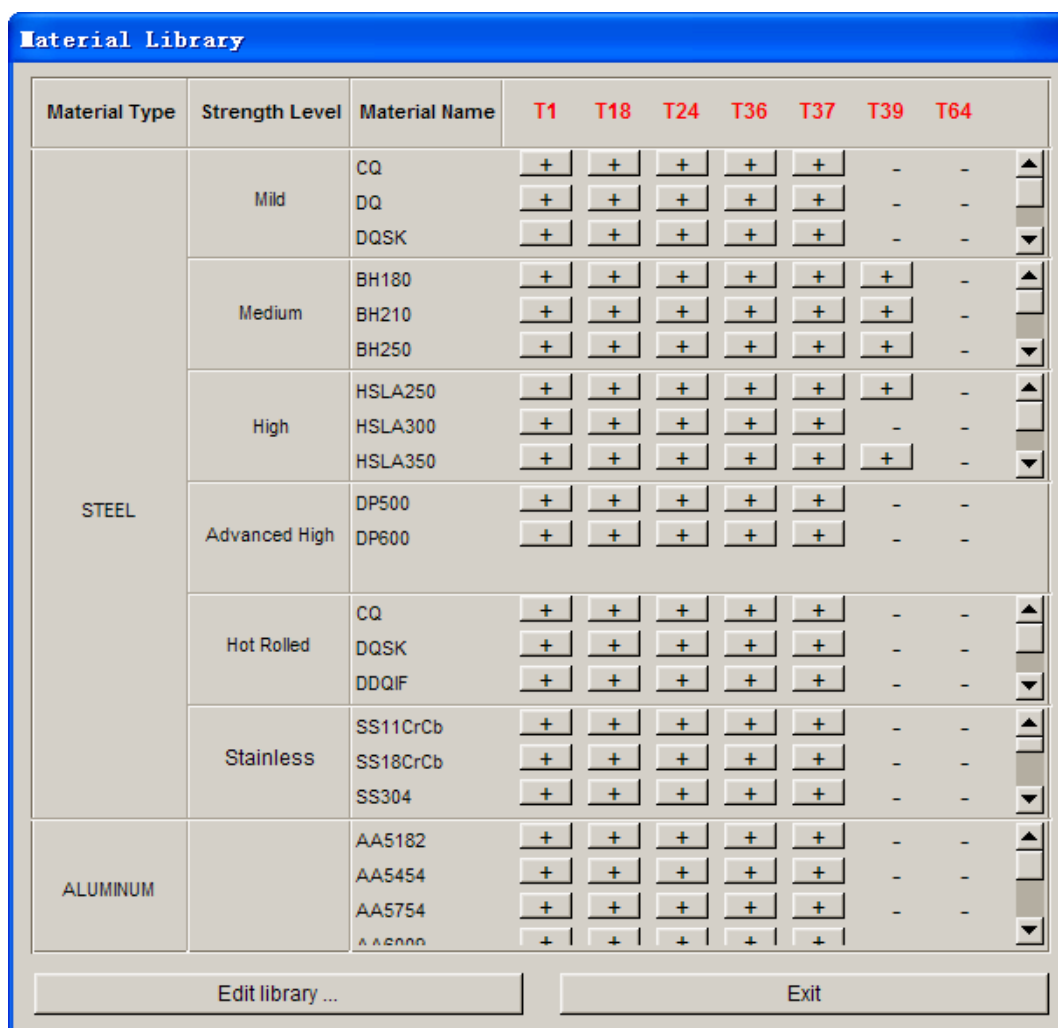


Figure 8.2.34 GUI of Material Library

**EDIT MATERIAL LIBRARY:** This function enables the user to edit the material library, such as Add, Delete and Modify material. After clicking this button, the program will display the Material Library Edit dialog box as shown in Figure 8.2.35.

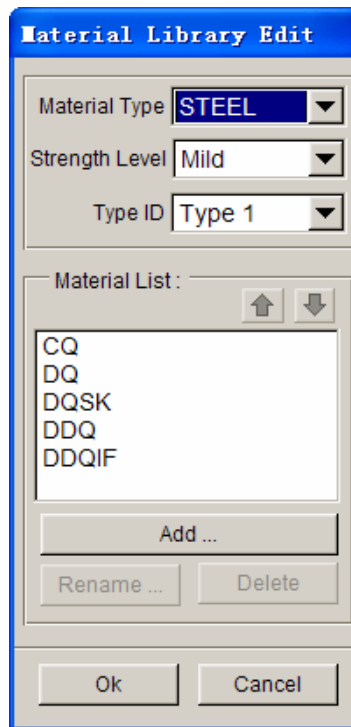



Figure 8.2.35 Edit Material Library dialog box

- **MATERIAL TYPE:** The user may switch the material type to select the corresponding material from material library. There are two types of materials in eta/DYNAFORM: Steel and Aluminum.
- **STRENGTH LEVEL:** The user can switch the material level to select the corresponding material from material library: Middle, Medium, High, Advanced High, Hot Roll and Stainless.
- **TYPE ID:** The user may switch the DYNA material type ID to select the corresponding material form material library. There are six material type ID: 1, 18, 24, 36, 37, 39 and 64.
- **ADD...:** Import a material file to the material library. The imported material will be added to the material library. Refer to the LS-DYNA User's Manual of for description of the material types.
- : The user needs to pre-select a material from the material list. Then, click those two buttons to change the position of the material in the list. Refer to the Figure 8.2.36 for an example.

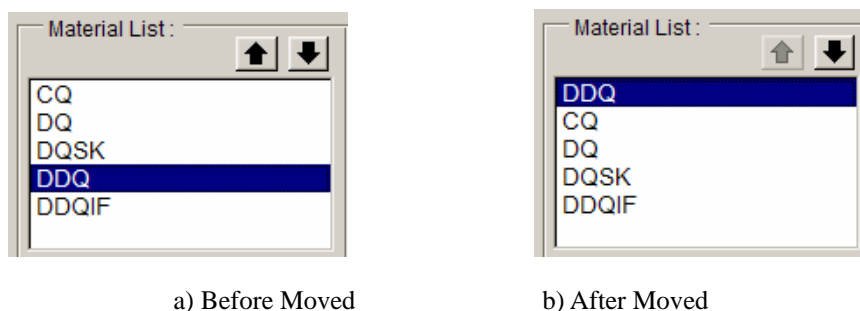


Figure 8.2.36 Change the position of material in material list

- **RENAME:** Enable the user to change the material name. The MODIFY NAME dialog box as shown in Figure 8.2.37 is displayed. The user may enter the new name in the edit box and click the OK button to accept the modification.

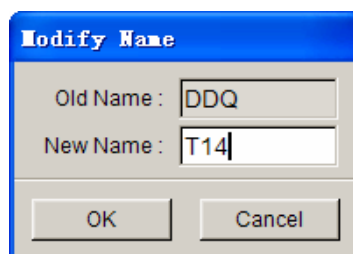


Figure 8.2.37 Modify Material Name dialog box

- **DELETE:** Enable the user to delete the selected material from material library.
- **OK:** Save the current setting and exit the Material Library Edit dialog box. The program will display a Question dialog box and prompt the user to overwrite the index as shown in Figure 8.2.38.

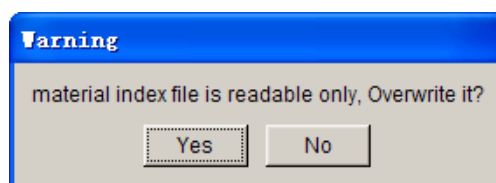


Figure 8.2.38 eta/DYNAFORM Question dialog box

- **CANCE:** Exit the Material Library Edit dialog box without saving the modification.

## THICKNESS

Under the default condition, all blank thickness is defined with default thickness in new setup. The user may specify different thickness for each part below the thickness bar.

## PROPERTY

The program specifies a default property for each defined blank part. The user does not need

to modify property under normal situation. The blank property includes selection of element formulation, selection of integration points number in blank thickness direction, and shear correction factor. See Figure 8.2.39 for the define property GUI window.

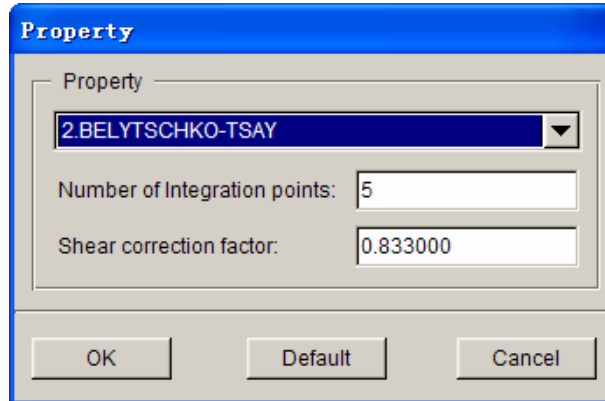


Figure 8.2.39 Define Property

**ELEMENT FORMULATION:** Allow the user to select an element formulation for shell element. eta/DYNAFORM provides 16 regular shell element formulations. The default element formulation is BELYSTSCHKO-TSAY. This formulation is the most commonly used and most efficient in current sheet forming simulation. For springback calculation, the user may select Full Integration element formulation.

**NUMBER OF INTEGRATION POINTS:** The default number of through thickness integration points is set as five. More through thickness integration points may result in accurate simulation. However, the simulation time is also increased drastically. Despite draw forming simulation with springback, the default 5 integration points are good for most cases. For springback simulation, 7 integration points is suggested as the best option.

**OK:** Save current modification and exit define property dialog box.

**DEFAULT:** Use default value for all parameter.

**CANCEL:** Exit dialog box without saving current modification.

#### 8.2.4.2 SYMMETRY

This function allows the user to add symmetry boundary condition or 1/4 symmetry condition to the blank. For completely symmetrical parts, the user may only define one half blank mesh and add symmetrical condition along the line of symmetry to save calculation time. For 1/4 symmetry parts, the user only needs to define the 1/4 blank mesh. Under default condition, the program considers the input as a whole blank. It displays NO SYMMETRY next to SYMMETRY TYPE, as shown in Figure 8.2.40.

If the user defines symmetry or 1/4 symmetry, the program will display the defined symmetry type in the dialog box.

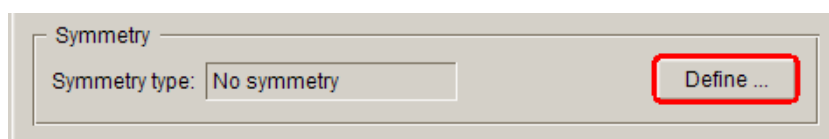


Figure 8.2.40 Define Symmetry

## 1/2 SYMMETRY



This function allows the user to define 1/2 symmetry blank. The GUI window is shown in Figure 8.2.41.

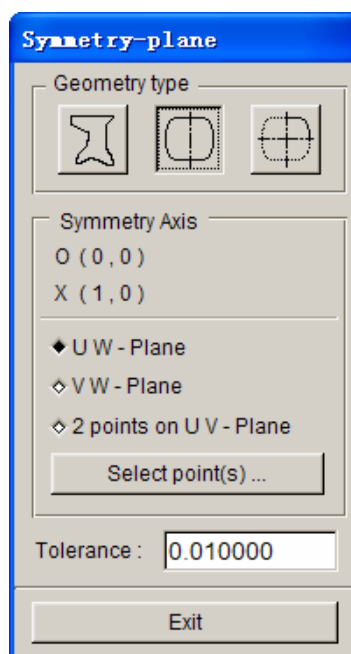


Figure 8.2.41 Define Symmetry Plane

**SYMMETRY AXIS:** This function displays the current user defined symmetry plane information. In new setup, the symmetry plane is specified by a point, a vector and WCS direction.

**O(0,0):** Indicate the required position of symmetry plane the vector starting point position for the defined plane.

**X(1,0):** Define the end point position of the vector for symmetry plane.

**UW-PLANE:** eta/DYNAFORM adopts a plane that is parallel to WCS's UW plane as

symmetry plane. The user may specify the plane location by using select points function. XY coordinate is displayed in O (0, 0), direction is displayed as X (1, 0). A vector is displayed in WCS's UW plane on the screen to show the symmetry plane. See Figure 8.2.42 for more details.

**VW-PLANE:** eta/DYNAFORM adopts a plane that is parallel to WCS's VW plane as symmetry plane. The user may specify the plane location by using select points function. XY coordinate is displayed in O (0, 0), direction is displayed as X (0,1).

**2 POINTS ON UV-PLANE:** Enable the user to select two points to define a vector. This vector forms a symmetry plane with WCS's W axis. The first point of the vector specifies the position of the plane.

**SELECT POINT(S):** Allow the user to select one or two points to specify symmetry plane. If the user selects UW-plane or VW-Plane, then only one point is required to specify the plane location. If the user selects 2-Points on UV-plane, then two points are required to define symmetry plane.

**TOLERANCE:** The user may adjust this value to search for nodes in the symmetry plane. After defining the plane, the program will change this tolerance to automatically search for the required nodes to add symmetry boundary condition. Default tolerance is 0.01. If there are nodes with larger tolerance from the defined symmetry plane, the user may adjust this value to extend the search range.

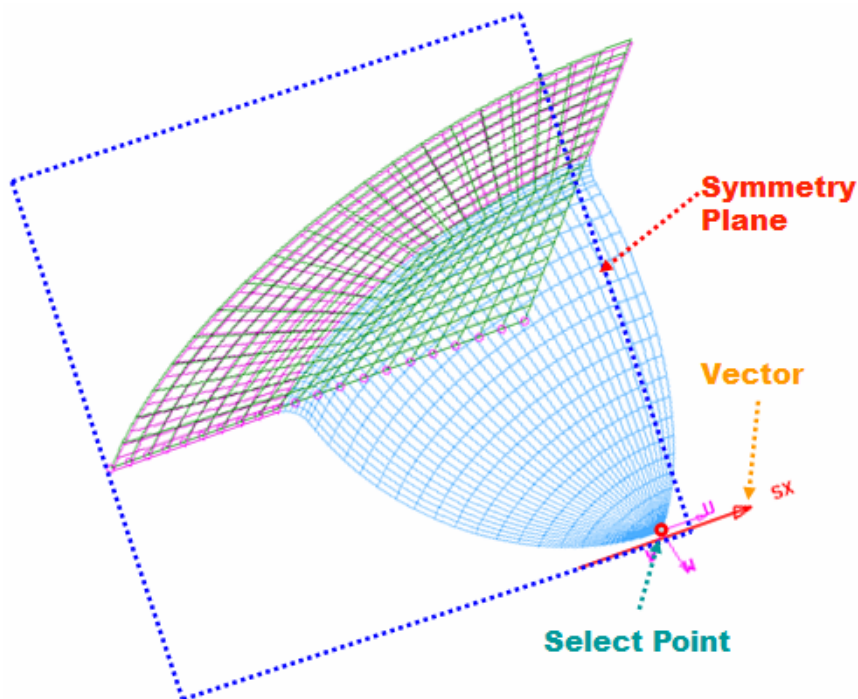


Figure 8.2.42 Define Symmetry

## 1/4 SYMMETRY

This function allows the user to add symmetry boundary for 1/4 symmetry parts. Its basic operation is same as 1/2 symmetry parts. After defining a vector, the program automatically generates a vector that is normal to the previous vector to specify another symmetry plane. See Figure 8.2.43 for more details.

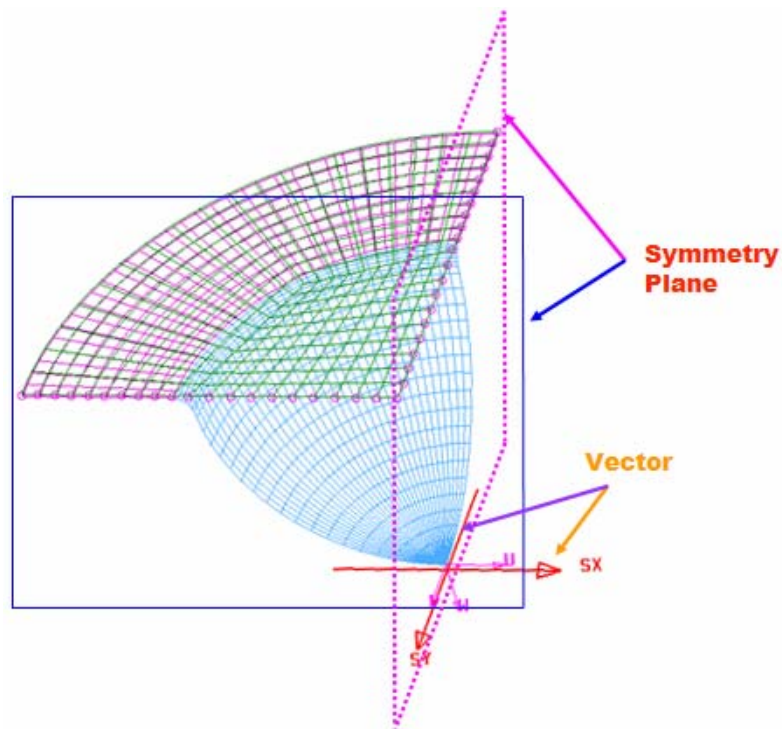


Figure 8.2.43 Define 1/4 Symmetry

### 8.2.4.3 WELDS

The user may conveniently set sheet forming of tailor welded blank and composite blank. After the user defined multiple blank parts, the weld option appears in Blank setup page, as shown in Figure 8.2.44.

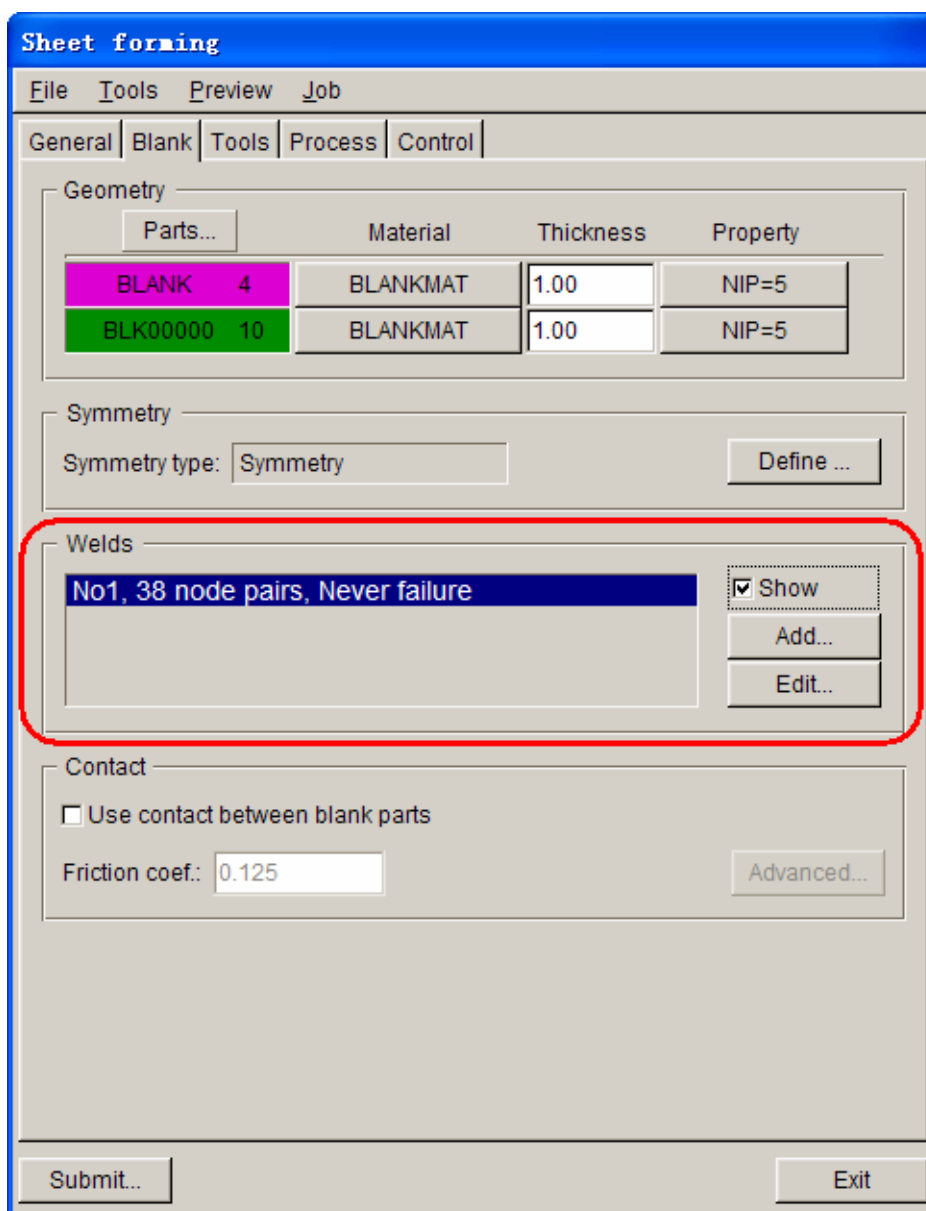


Figure 8.2.44 Welded Blank Define

If the user has already defined several welding seams, information of welding seams will be displayed in welds list.

**ADD...**

This function allows the user to select adjacent nodes to define weld element. After clicking this button, the DEFINE BLANK WELD dialog box, as shown in Figure 8.2.45, is displayed.

*NOTE: In LS-DYNA, the defined node pair for weld must be two different nodes at different locations. Different nodes at same location or identical node are not allowed as node pair. In order to reduce the preparative time for user, eta/DYNAFORM will separate automatically the*



*identical nodes which are used to define the weld. User only needs to add the corresponding elements to different parts.*

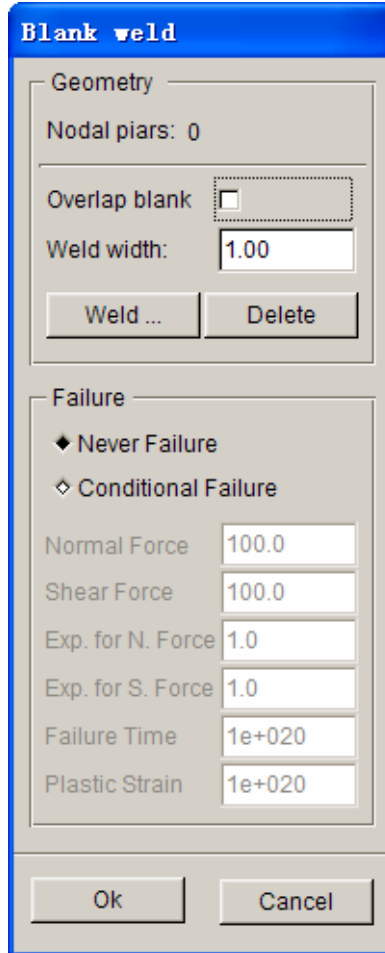


Figure 8.2.45 Define Weld dialog box

**NODAL PAIRS:** Display number of nodal pairs. After the user selecting nodal pairs to define weld, the number is displayed in the data box.

**OVERLAP BLANK:** The user needs to select this option for overlap blank. After selecting this option, the line connecting the nodal pair must be normal to its connecting element.

**WELD WIDTH:** Specifies weld width. Default width is 1.0 (mm). If the distance between selected node pair is smaller than this width, the program will define weld for the node pair. The user may adjust the weld width to include the required nodal pairs with greater distance.

**WELD...:** The user may click this button to select nodal pair. After clicking this button, the program display the select nodal pair dialog box. The user may select two rows of nodes for required weld line. After selecting the nodes, the program will

automatically define weld on nodal pairs, and highlight the welds as shown in Figure 8.2.46.

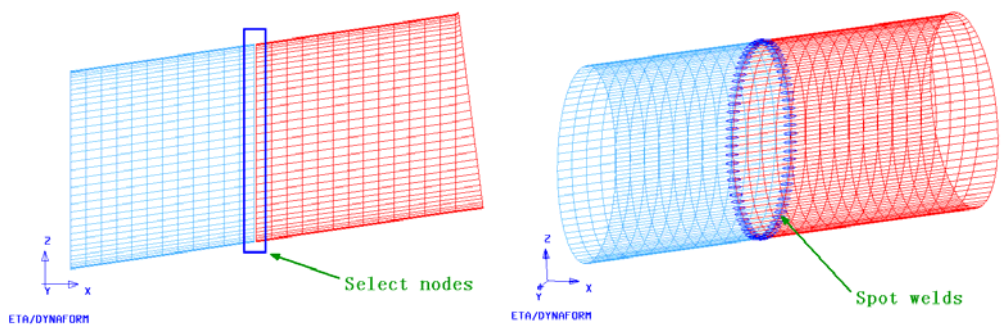


Figure 8.2.46 Weld Define

If there is only one row of nodes, the program will separate the node and offset it automatically as shown in Figure 8.2.47.

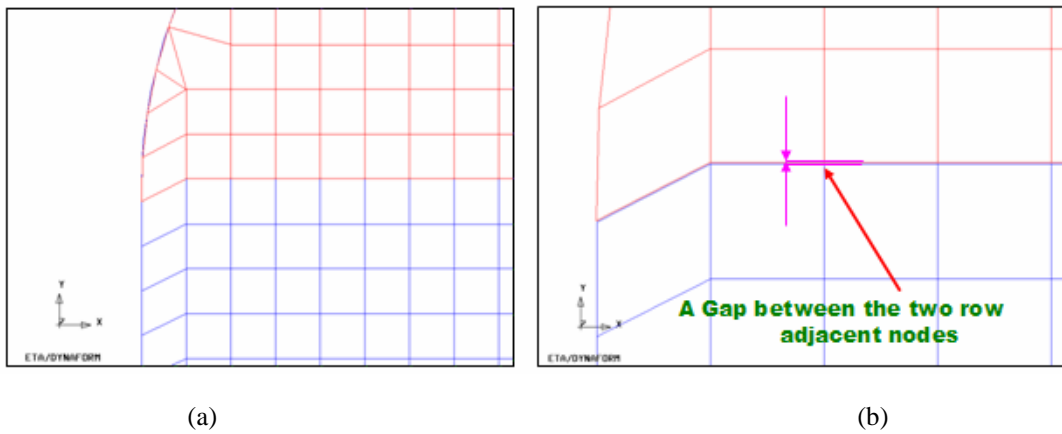


Figure 8.2.47 Node Offset (a) Before (b) After

**DELETE:** This button allows the user to delete all current nodal pairs.

**FAILURE:** The user may select failure criteria for the weld.

**NEVER FAILURE:** Under default condition, select NEVER FAILURE option to enable no failure at the welds during the simulation.

**CONDITIONAL FAILURE:** Allow the user to define conditional failure criteria and the relevant parameters. Detailed description is available in LS-DYNA User’s Manual.

**EDIT**

This function allows the user to edit the defined weld. Similar to define weld, the user may delete selected nodal pair, add new nodal pair, modify weld failure criteria, etc.

**DELETE**

After defining a weld line, information of the weld is displayed in the weld list. It includes

number of nodal pairs, failure rule, as shown in Figure 8.2.48. The user may delete a weld by clicking the Edit button and the Delete button to delete the current weld.

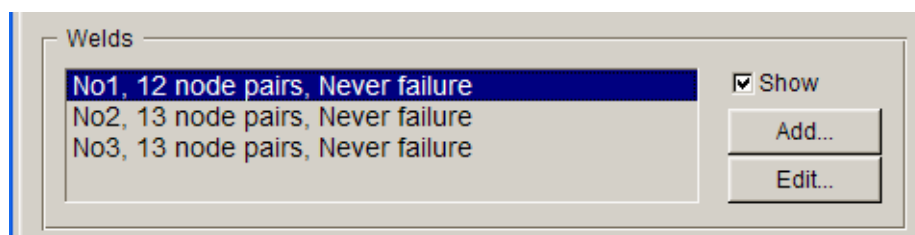


Figure 8.2.48 Welds List

#### 8.2.4.4 CONTACT

For overlap blanks, the contact interface between blank parts has to be defined. The user may modify contact parameters, such as friction coef., contact coef. type, etc. as shown in Figure 8.2.49. For general welded blank, it is not unnecessary define the contact between the blank parts.

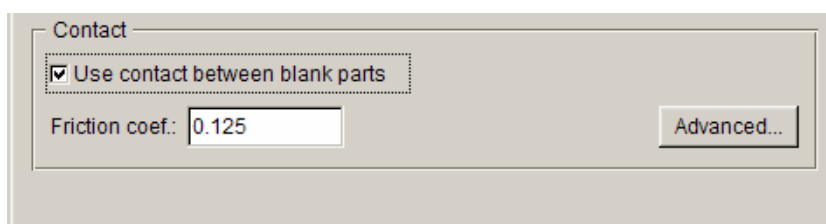


Figure 8.2.49 Define Contact Between Blanks

#### USE CONTACT BETWEEN BLANK PARTS

This option is only necessary for overlap blanks.

#### FRICITION COEF

Define friction coefficient between blank parts. Default friction coefficient is 0.125.

#### ADVANCED...

The user may modify some contact parameters between blanks through advanced options as shown in Figure 8.2.50.

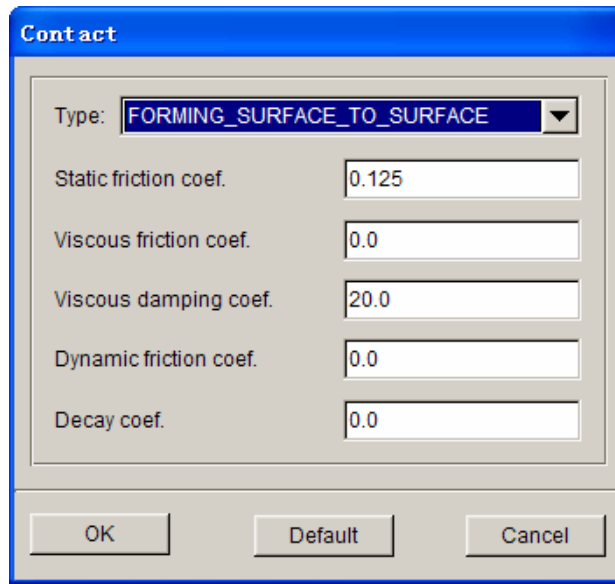


Figure 8.2.50 Define Contact Advanced Parameter

**CONTACT TYPE:** This function allows the user to select contact type. eta/DYNAFORM includes commonly used contact types for the blank.

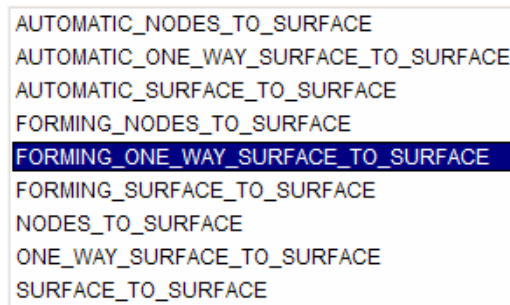


Figure 8.2.51 Define Contact Type

**STATIC FRICTION COEF:** Allow the user to define static friction coefficient. Default value is 0.125.

**VISCOUS FRICTION COEF:** Allow the user to define viscous friction coefficient. Default value is 0.

**VISCOUS DAMPING COEF:** Allow the user to define viscous damping coefficient. Default value is 20.0.

**DYNAMIC FRICTION COEF:** Allow the user to define dynamic friction coefficient. Default value is 0.

**DECAY COEF:** Allow the user to define decay coefficient. Default value is 0.

## 8.2.5 TOOLS

In the Tools definition page, the user may add and delete tools, positioning between tools, define tool part, working direction and contact between tool and blank. The main interface is shown in Figure 8.2.52.

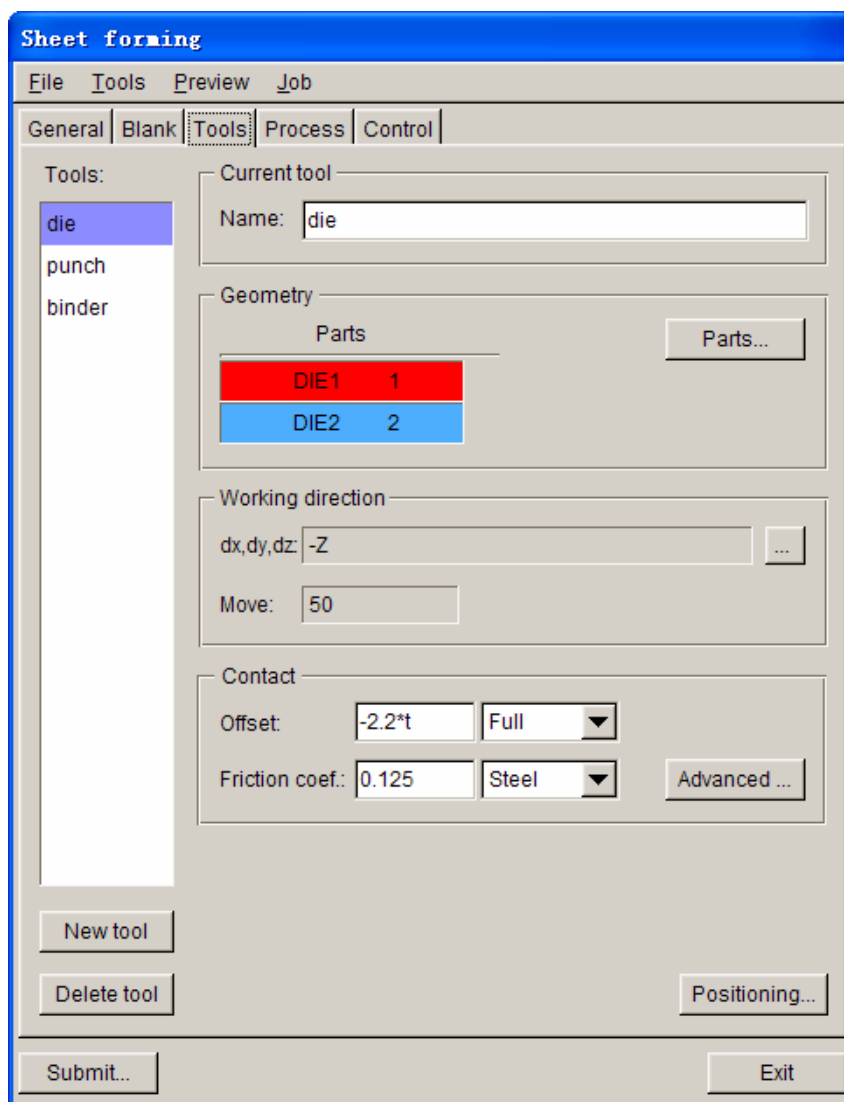


Figure 8.2.52 Tools definition page

### TOOL LIST

On the left side of Tools definition page, there is a list of defined tool names. The highlighted tool name is the working tool. The user can mouse click the tool name from the list to switch current tool. During a new setup, the program sets default tool according to the user selected process type. For example, if the user selects SINGLE ACTION, the program creates three default tools, which are die, punch and binder. Every basic parameter is set to default, except for tool part, working direction, contact offset, etc. The user may define each tool part directly from the menu page, which greatly saves the tool

definition time.

*NOTE: The new tool is listed in red, which indicating the definition of tool parameters is not completed. In new setup, the user must define every necessary parameter. Otherwise, the title of the tool page is displayed in red to remind the user that there is incomplete setup.*

### 8.2.5.1 NEW TOOL

The NEW TOOL function allows the user to create a new tool in current setup. The user may click NEW TOOL button to switch the interface to the new tool interface, as shown in Figure 8.2.53.

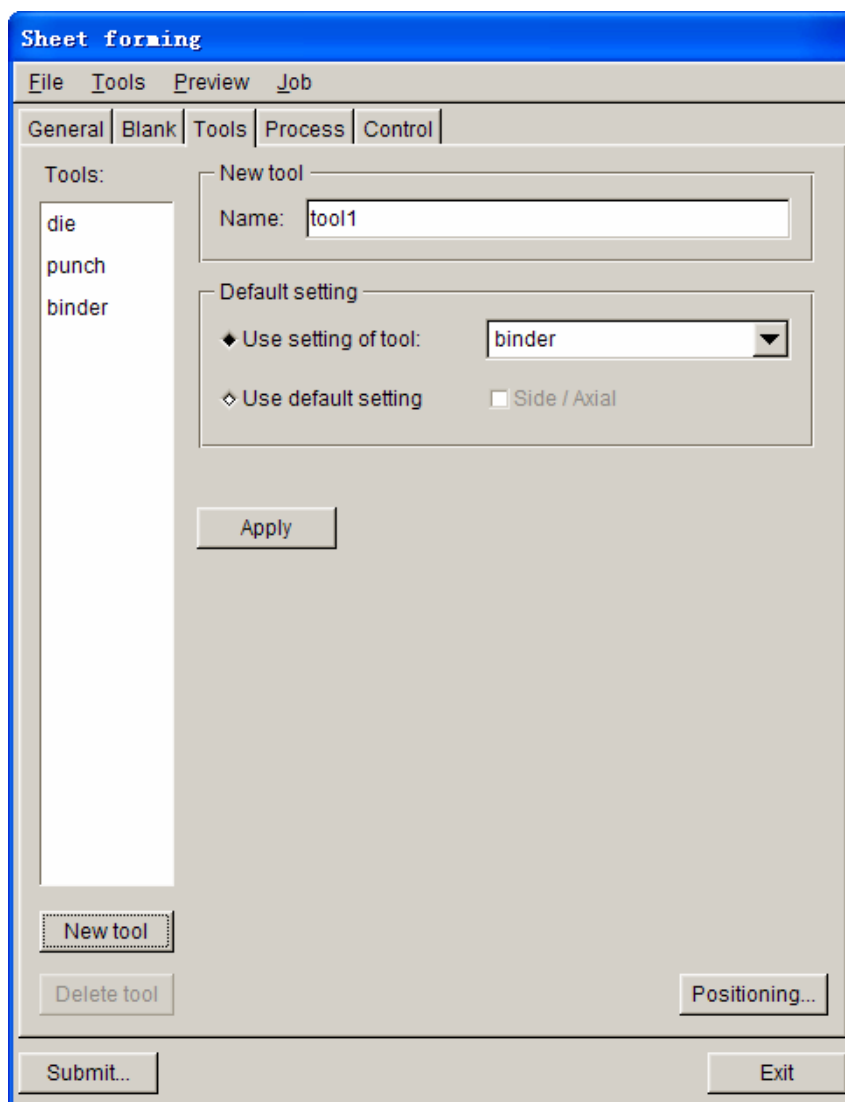


Figure 8.2.53 New Tool interface

### TOOL NAME

Allow the user to define a name for the current tool. Default tool name is tool1, tool2, tool3, etc. The user may define a meaning full name for the current tool for the ease of identifying each tool in the

stamping simulation.

## DEFAULT SETTING

Allow the user to set defaults for the current tool. It includes working direction, offset, etc. There are two options in default setting. One is to take existing tool as reference, set basic parameter of newly generated tool as in the selected existing tool. The other one is to set the basic parameter of the newly generated tool as default value.

**USER SETTING OF TOOL:** Use an existing tool's parameters for the new tool. The user may select an existing tool from the pull-down list. The parameters of two tools are basically the same. Only minor modification to the parameters is necessary for the newly generated tool.

**USER DEFAULT SETTING:** Use system default parameter to define the new tool. The working direction of this tool is WCS +W direction. Both contact offset and travel distance is equal to 0.

**SIDE/AXIAL:** The user needs to define the contact type of new tool and blank. There are two different contact types in the auto setup. The first type is normal axial contact. For example, contacts of punch and blank, die and blank are of this type. The other type is side contact. Contact of location (guide) pin and blank, blank and blank, hydro forming side and blank are of side contact. The user needs to determine the type of contact to be used between current tool and blank.

## APPLY

The user may click this button to generate new tool. Newly generated tool is added in the tool list.

### 8.2.5.2 DELETE TOOL

This function allows the user to delete the current tool. The user may select a tool from tool list to be the current tool, and click the DELETE TOOL button. The program will display a confirmation dialog box as shown in Figure 8.2.54.

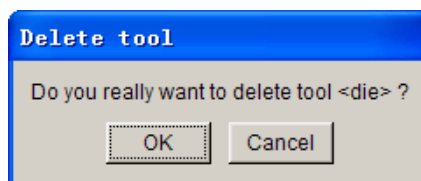


Figure 8.2.54 Delete Tool dialog box

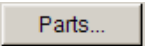
Click OK to delete the current tool, or Cancel to cancel the operation.

**8.2.5.3 TOOL NAME**

This function allows the user to modify the current tool name. When creating a tool, the user may assign a name for the tool. After the tool is generated, the user may change the tool name in tool define main interface. In the NAME bar, the user may enter new name and press “Enter [CR]” to confirm.

*Note: Every tool must have a unique name.*

**8.2.5.4 TOOL GEOMETRY**

This function allows the user to define parts for the current tool. The user may add one or more parts to the current tool. Click  button to display the DEFINE GEOMETRY dialog box as shown in Figure 8.2.55.

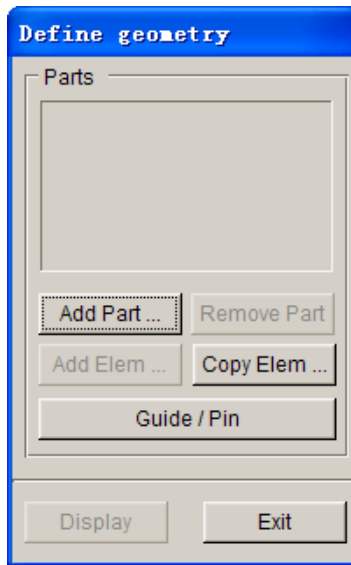


Figure 8.2.55 Tool Define Geometry dialog box

**ADD PART...:** This function allows the user to select one or more parts from part list to define the tool. The selected parts are displayed in Pars List window. See Chapter 4 for description about selecting parts.

**REMOVE PART:** This function allows the user to remove selected parts form the Part List.

**ADD ELEM...:** This function allows the user to add selected element(s) to selected part.

**COPY ELEM...:** This function allows the user to COPY or OFFSET new element(s) form existing elements. If the user defines only one die face, then he/she may COPY or OFFSET new parts by using this function, and define the part as the relevant tool. The program displays the copy element dialog box as shown in Figure 8.2.56.



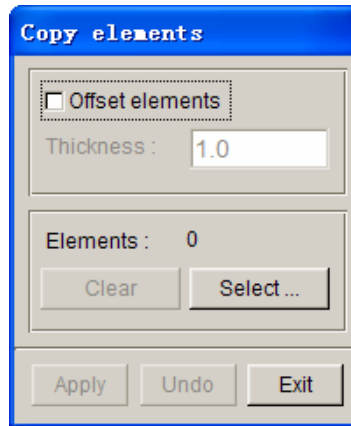


Figure 8.2.56 Copy Elements dialog box

**OFFSET ELEMENTS:** This function allows the user to offset new elements from existing elements to a new part in the current tool. The name of the new part is OFFSET01, OFFSET02, etc.. Under normal condition, this option is not selected, which indicates only copy element operation is conducted. Newly generated element is at the same location as the original element.

**THICKNESS:** Element offset thickness. Default thickness is 1.0 (mm). The user may change this value to the required offset distance. The user may offset the elements to the opposite direction by setting a negative value, such as -1.0. The program will offset new elements in the opposite direction of the element normal.

**SELECT...:** This function allows the user to select element(s) to copy and offset. After clicking this button, the program displays the select element dialog box.

**CLEAR:** This function allows the user to clear selected elements. The user may use SELECT button to select elements again.

**APPLY:** After selecting elements, the user may click APPLY to copy and offset.

**UNDO:** The user may click UNDO to cancel the last copy and offset operation.

**EXIT:** Exit copy element dialog box.

**GUIDE/PIN:** This function allows the user to create GUIDE and/or PIN, and create a new part for the GUIDE as current tool. Default name of new GUIDE is ACC0000, ACC0001...and so on. For detailed description of GUIDE and PIN, refer to Section 10.6.

**DISPLAY:** This function allows the user to display parts that are corresponding to the current tool on the screen. After user defined tool geometry, parts that are corresponding to this tool are displayed in

the part list.

**8.2.5.5 WORKING DIRECTION**

In the new auto setup, a unique working direction has to be defined to every tool. Working direction is the moving direction from the tool toward the blank. Note: working direction is a vector, starts from the tool and points to the blank. It is the line of the tool’s movement. It is not necessarily the actual moving direction of tool. It can be opposite to the moving direction. The interface of working direction is shown in Figure 8.2.57.

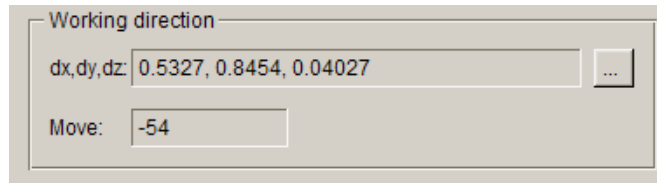



Figure 8.2.57 Define Working Direction

**DX, DY, DZ:** This function displays the current tool working direction as a vector. Default starting point of the vector is WCS (0, 0, 0). The coordinate of the end point is also displayed in this box. The

User may click  button to change the working direction of this tool. After clicking this button, the program displays the DEFINE DIRECTION dialog box as shown in Figure 8.2.58. Detailed descriptions of define direction is provided in Section 8.2.3.2.

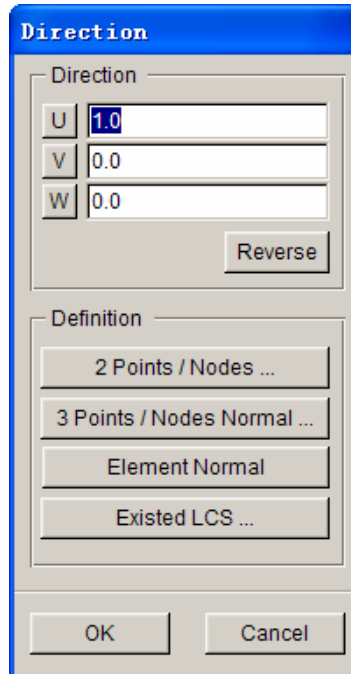


Figure 8.2.58 Define Tool Working Direction

**MOVE:** This function displays the tool travel distance from tool’s original position after positioning the current tool. This value is the travel distance of the tool along the working direction. If it is negative,

the tool moves along the opposite direction of working direction.

## CONTACT

This function allows the user to define contact parameter between the blank and tool.

**OFFSET:** When the user defined tools are copied from the same die face, contact offset is required for corresponding tools. For example, if the mesh is generated according to die face, an offset value is required for the corresponding punch and binder ring. The blank thickness is used for the offset value. The user may change the value according to actual requirement.

**FRICTION COEF:** Allow the user to specify friction coefficient of tools.

**ADVANCED:** Define advanced parameters for contact interface. Detailed description of contact definition is available in Section 8.2.4.4 and LS-DYNA User's Manual.

## POSITIONING...

This function is used to control the initial position of tooling and blank in the stamping simulation, relatively to the original position. This function includes auto positioning and manual positioning as shown in Figure 8.2.59. A reference tool is required for auto positioning. It is fixed during positioning and acts as reference for the positioning of other tools and blanks.

Positioning	
Blank	
Translation:	0
Tool	
Tools	Movement
die	0
punch	0
binder	0
Auto-position	
Fixed:	punch
<input checked="" type="checkbox"/> Round off	Auto
Reset	OK Cancel

Figure 8.2.59 Tool Positioning

**BLANK TRANSLATION:** The user may enter a value in the input box to translate the blank.

If the user chooses auto positioning, the translation of blank from its original position is displayed in this dialog box.

**TOOL MOVEMENT:** The program lists all define tools in the tool bar. The user may define initial position of tooling by entering a value for each tool in the input data field. If the user selects auto positioning, distance from tool to its original position is displayed in this dialog box.

**NOTE:** *Auto positioning of tool and blank are both along working direction. The user must specify working direction for each tool prior to performing auto-positioning.*

**AUTO-POSITION:** This function allows the user to auto-position the initial position of tooling. A fixed tool during actual forming process should be selected as the reference tool. For example, the punch is selected as reference tool for single action forming process. For double action forming process, the die is selected as reference tool. After auto positioning, distance of every tool from their original position is displayed in input box. The user may adjust the value as required.

**NOTE:** *The minimum gaps between the blank and tools are equal to the thickness of blank in the Auto Position. If multiple blanks are defined with different thickness, the largest thickness is selected.*

**ROUND OFF:** This function allows the user to control round off of auto positioning value of the tool or blank. If this function is selected, the program sets every tool travel distance a rounded integer number after auto positioning.

**OK:** Click this button to save the positioning result to database and exit positioning dialog box.

**EXIT:** Click this button to exit the positioning dialog box without saving positioning result.

## 8.2.6 PROCESS

The PROCESS definition page includes the basic setup of process parameters. It includes stamping speed, closing force, forming time, etc. The user may change process parameters according to the actual condition. Moreover, the user may add and delete any process. The basic process setup interface is shown in Figure 8.2.60.

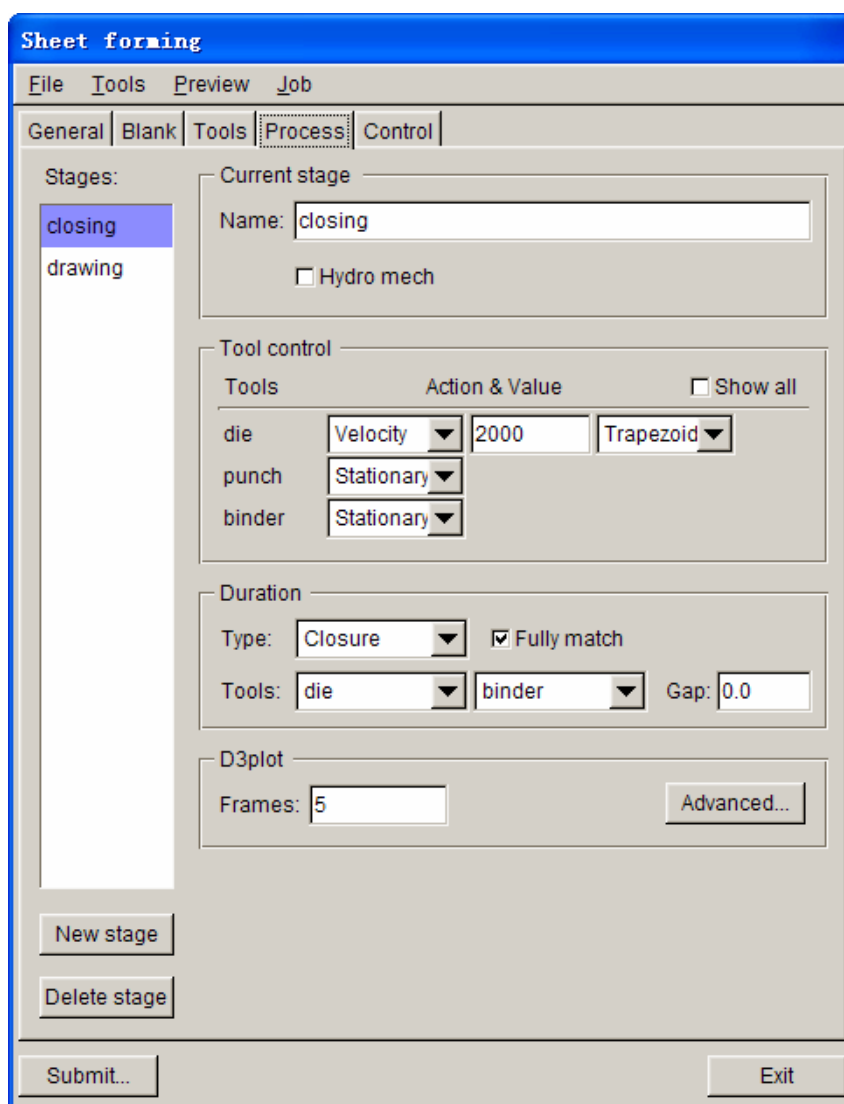


Figure 8.2.60 Process definition page

## PROCESS LIST

On the left of process definition page, all defined process names are listed in the window of Stages. The current process name is highlighted in the list. The user may click another process from to list to switch the current process. The program specifies default process according to the user selected process type at the beginning of the new setup. For example, if the user selects SINGLE ACTION, the program will create closing and drawing as default process. There are also default settings for the basic parameters, including stamping speed, closing power and stage calculation time, etc. The user only needs to change a few parameters as necessary for the analysis.

### 8.2.6.1 NEW STAGE

This function allows the user to add a new process in current setup. The user may click NEW STAGE button to switch to a new stage interface, as shown in Figure 8.2.61.

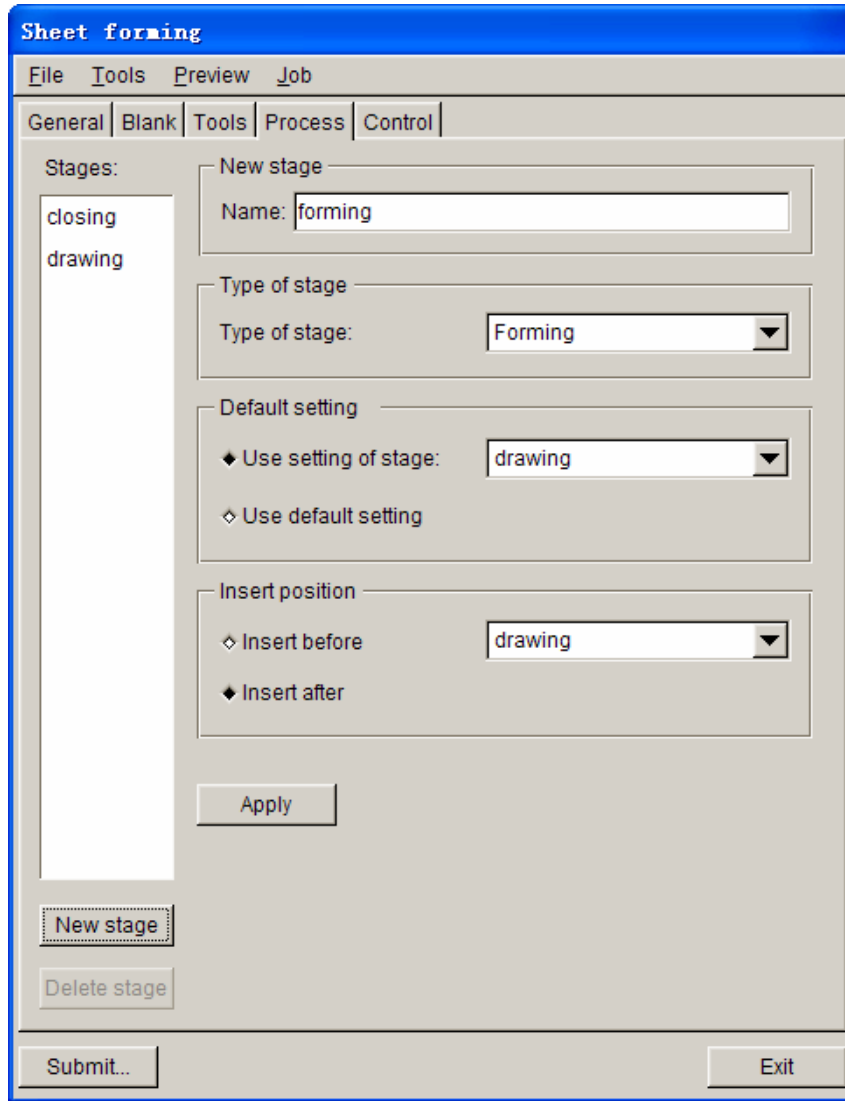


Figure 8.2.61 New Stage

**STAGE NAME**

Allow the user to define a name for the current stage. Default stage name are forming1, forming2, forming3, etc.. The user may define a meaningful name for current stage, to make it more convenient to understand, setup and check faults.

**TYPE OF STAGE**

Allow the user to specify a type for current stage. The user may select from the pulling down list. Stage types include normal forming and springback.

**DEFAULT SETTING**

Allow the user to select default setting for the new stage. In default setting, the user may select the

program default parameters for the new stage. Otherwise, the user may select to use the parameters from a defined stage for the new stage.

**USER SETTING OF STAGE:** Use an existing stage's parameters for the new stage. The user may select an existing stage from pulling down list. The new stage will have the same parameters as the selected stage. The user only needs to make few changes as necessary for the new stage.

**USER DEFAULT SETTING:** Use system default parameters to define a new stage.

## INSERT POSITION

Allow the user to specify the insert position of the new stage. Since the stages have to follow the order in actual forming process, the user needs to specify stages' position accordingly.

**INSERT BEFORE:** Insert current new stage before a specified stage from the pulling down list.

**INSERT AFTER:** Insert current new stage after a specified stage from the pulling down list.

### 8.2.6.2 DELETE STAGE

This function allows the user to delete the current stage. The user may select a stage from the stage list as current stage, and click DELETE STAGE button. The program will pop up a confirmation dialog box as shown in Figure 8.2.62.

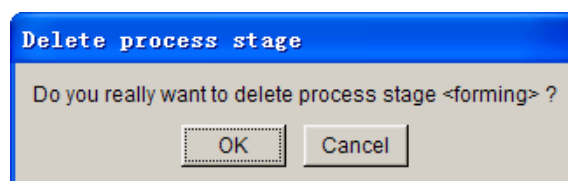


Figure 8.2.62 Delete Stage dialog box

Click on OK button to delete the current stage or Cancel button to cancel the delete operation.

### 8.2.6.3 STAGE NAME & TYPE

#### NAME

Display the current stage name. The user may type in a new name in the name window.

**TYPE**

Allow the user to select the current stage type: Drawing or Hydroforming.

**8.2.6.4 TOOL CONTROL**

This function is used to control tools' action in the current stage. It includes define travel speed, closing force and travel curve, etc. of the tools. See Figure 8.2.56 for a sample tool control.

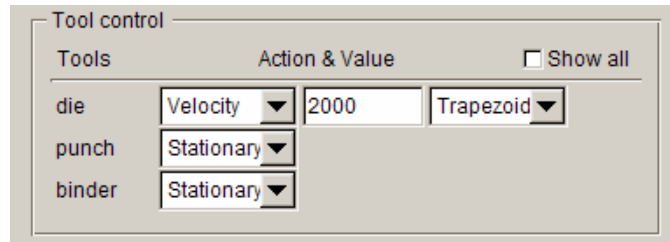


Figure 8.2.63 Tool Control Option

**TOOLS**

List all defined tools in current stage. Under default condition, the non-active tools in the current stage are not listed. The user may select SHOW ALL option to show all defined tools and activate the tools accordingly.

**SHOW ALL**

List all defined tools in tool list. This option is not selected under default condition.

**ACTION & VALUE**

This group lists all tool control options. There are five control methods of each tool: NON-ACTIVE, STATIONARY, VELOCITY, DISPLACEMENT and FORCE.

**NON-ACTIVE**

Indicate this tool is not used in this stage. Under default condition, not-active tools are not displayed in the tool list. The user may select SHOW ALL option to display all tools.

**STATIONARY**

Indicate this tool is stationary in this stage.

**VELOCITY**

Indicate the tool is controlled by velocity curve in this stage. The user may select one of the standard types provided in eta/DYNAFORM to define velocity curve: TRAPEZOIDAL, SINUSOIDAL, SIN.W-H, TRIANGULAR and VARIABLE. Figure 8.2.64 shows the standard curves. The user may enter the maximum velocity value in velocity input box. Default velocity for drawing process is set as 5000mm/s. For closing process, default velocity is set as 2000mm/s.



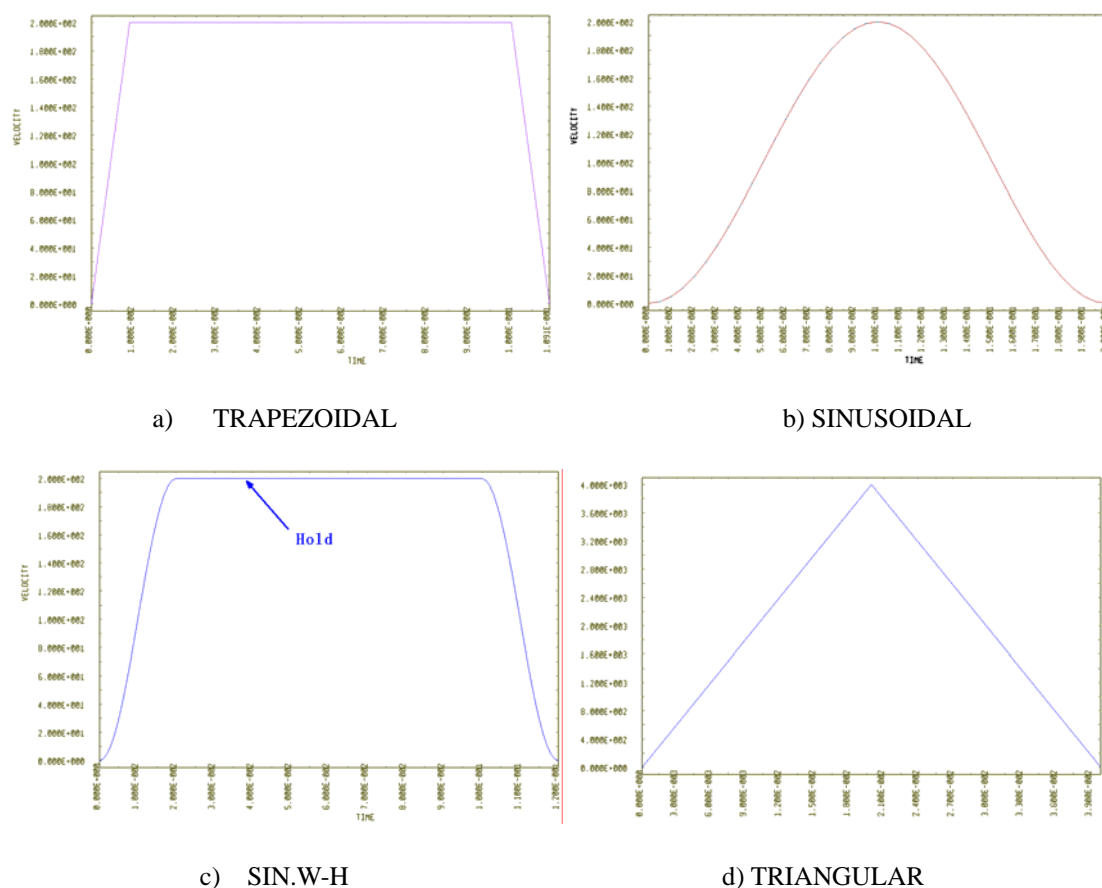



Figure 8.2.64 Different Types of Curves

If the user selects user-define curve,  button is activated to enable the user to define the curve in the define curve dialog box.

### DISPLACEMENT

Use displacement-time curve to control the tool travel in this stage. The user may click DEFINE button to define displacement-time curve. The procedure is similar to those described in the velocity curve.

### FORCE

Use force to control the tool operation in current stage. Force is usually used for ring closure. There are two types of forces: CONSTANT and VARIABLE. If the user selects constant force, a constant force value is required. For variable force, the user needs to click DEFINE to define force-time curve in the define curve interface.



This button is activated when the user selects force control. The user may use this button to define RIGID BODY STOPPER. Rigid body stopper is mainly used when using force to control the binder ring to prevent lack of closing force that causes binder ring pop away or excessive closing force that restricts the metal flow. The user may specify the parameters for

rigid stopper as shown in Figure 8.2.65.

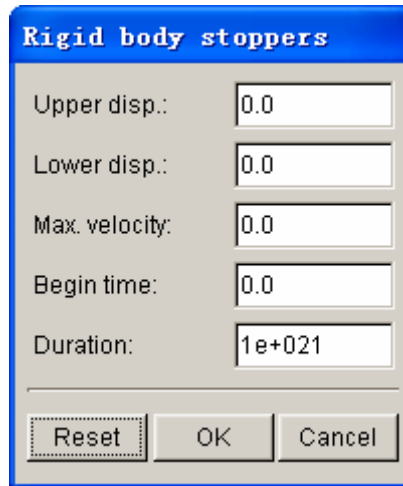


Figure 8.2.65 Rigid body stoppers dialog box

**UPPER DISP.:** The limit of upper maximum displacement of the tool from its initial position.

**LOWER DISP.:** The limit of lower maximum displacement of the tool from its initial position.

**MAX. VELOCITY:** The maximum velocity as function of time that is allowed for the tool with rigid body stopper control during the process.

**BEGIN TIME:** RIGID STOPPER starts to work.

**END TIME:** RIGID STOPPER termination time.

**DURATION:** RIGID STOPPER duration time.

After completing RIGID STOPPER definition, the button's colour turns to blue. Detailed description of RIGID STOPPER is available in LS-DYNA User's Manual.

### 8.2.6.5 USER DEFINE CURVE

In the new setup, all the defined curves are managed by the curve manager. The user may create, import, modify existing curve by the curve manager. Curve manager's main interface is shown in Figure 8.2.66.

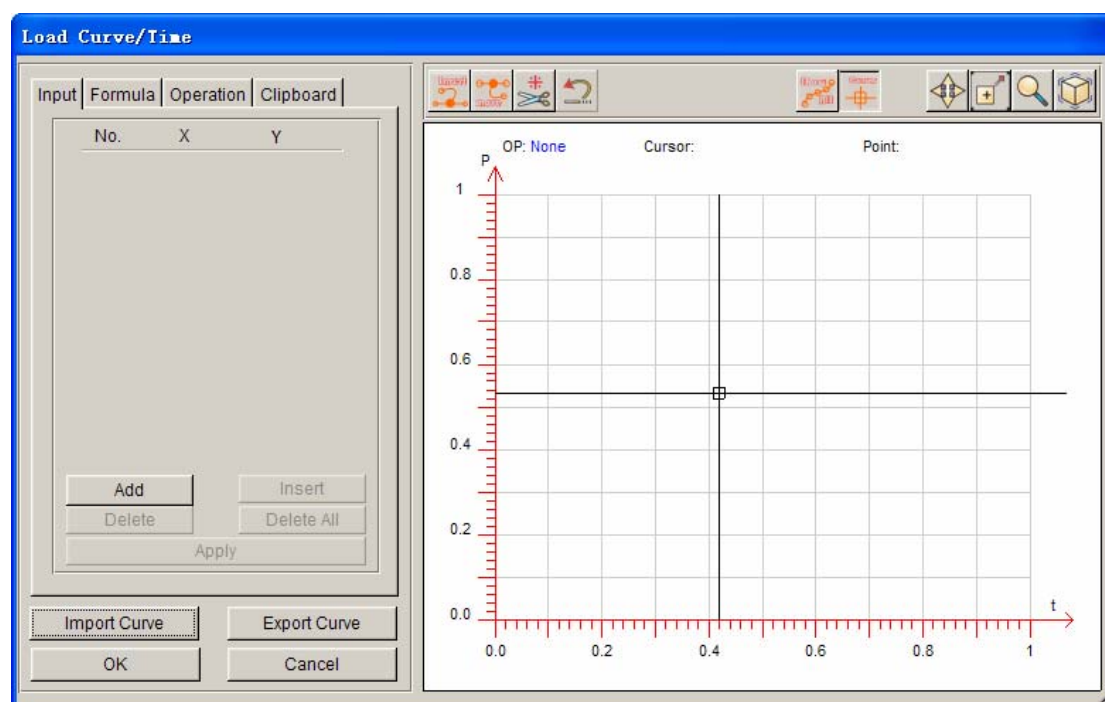


Figure 8.2.66 Curve Manager

### IMPORT CURVE

Allow the user to import a curve from a file. The extension of curve files is “.cur”. The curve file is defined according to the standard LS-DYNA keyword format. A typical “.cur” curve format is shown below. After clicking this button, the user needs to select a “.cur” file from the dialog box.

```

$-----1-----2-----3-----4-----5-----6-----7-----8
$
$           ETA/DYNAFORM USER CURVE
$
$-----1-----2-----3-----4-----5-----6-----7-----8
*DEFINE_CURVE
$CURVETYPE: TIM_CURVE
$   LCID   SIDR   SCLA   SCLO   OFFA   OFFO
$         101     0
$
$           A1           O1
$           0.0000      0.0000
$           0.1000      0.1100
$           0.2000      0.2400
$           0.3000      0.3900
$           0.4000      0.5600
$           0.5000      0.7500
$           0.6000      0.9600
$           0.7000      1.1900
$           0.8000      1.4400
$           0.9000      1.7100
$           1.0000      2.0000
$           1.1000      2.3100
$           1.2000      2.6400
$           1.3000      2.9900
$           1.4000      3.3600
$           1.5000      3.7500
$           1.6000      4.1600
$           1.7000      4.5900
$           1.8000      5.0400
$           1.9000      5.5100
$           2.0000      6.0000
$-----1-----2-----3-----4-----5-----6-----7-----8
*end

```

### EXPORT CURVE

Allow the user to export current curve to a “.cur” file. The user needs to specify a file name after clicking this button. The program automatically saves the current curve under this name and directory.

### INPUT

This page displays coordinates of the points on the current curve. If there is no existing curve, user may create a new curve by entering point coordinates. The user may modify existing curve coordinate in this page, including functions of add, delete, insert and clear of all data points.

**ADD:** Allow the user to add a data point at the end of curve. The program adds two empty input boxes at the end of point list, and automatically specifies the data point as the current point. The point is highlighted in yellow color. The user may enter X and Y coordinates in the input boxes.

**INSERT:** Allow the user to insert a data point after the current data point. Before clicking this button, the user needs to define the position of the current data point. Click the data point where the new data is inserted below. The program will highlight the selected data point. After clicking the INSERT button, the program automatically inserts two empty input boxes below the highlighted data point. Enter the X and Y coordinates in the input boxes for the new data point.

**DELETE:** Allow the user to remove the current data point.

**CLEAR ALL:** Allow the user to clear all data points in the current curve.

**APPLY:** The user may click APPLY after add, insert or delete point operation. The program automatically displays the entire curve in the curve display window after the modification.

### FORMULA

This page allows the user to use some common mathematic formula, such as linearity, quadratic, cubic, sin (cos) to create a curve. When defining FLC curve of material, the user may create a curve according to the traditional Keeler’s empirical formula.

The screenshot shows a software window titled 'Formula Page'. It has four tabs: 'Input', 'Formula' (which is active), 'Operation', and 'Clipboard'. In the 'Formula' tab, there is a 'Type:' dropdown menu currently set to 'Power law'. Below this, there are three input fields: 'Interval' with the value 0.02, 'Absci-start' with the value 0.02, and 'Absci-end' with the value 1. A central box displays the formula  $\sigma = \sigma_y + K(\epsilon)^n$ . Below the formula are three more input fields:  $\sigma_y$  with the value 274,  $K$  with the value 648, and  $n$  with the value 0.22. At the bottom of the window is an 'Apply' button.

Figure 8.2.67 Formula Page

**TYPE:** Allow the user to select a math formula type to create a curve. Type of formula includes SIMPLE, QUADRATIC, CUBIC, SIN and COS.

**INTERVAL:** Allow the user to specify interval of selecting X coordinate points during curve creation. The smaller the value is, the more data points will be required to define the curve. Default interval is 0.1.

**ABSCI-START:** Allow the user to specify ABSCI- START during curve creation. Default ABSCI-START is 0.

**ABSCI-END:** Allow the user to specify ABSCI-END during curve creation. Default ABSCI-END is 20.

**PARAMETER:** Allow the user to input required formula parameter during curve creation. After the user selected one formula type, the expression of the formula is displayed below the interface. For example, formula of QUADRATIC is:  $y = Ax^3 + Bx^2 + Cx + D$ . To create curve, the user needs to specify quadratic curve required four parameters A, B, C and D for the point. Under default condition, each formula parameter is set to 1.

**APPLY:** After defining formula parameters, the user may click APPLY button to create the curve. The created curves are displayed in show curve window. At the same time, coordinates of the data points on the curve are displayed in the INPUT page.

## OPERATION

This page allows the user to operate on the entire curve, such as SCALE, TRANSLATE, etc.

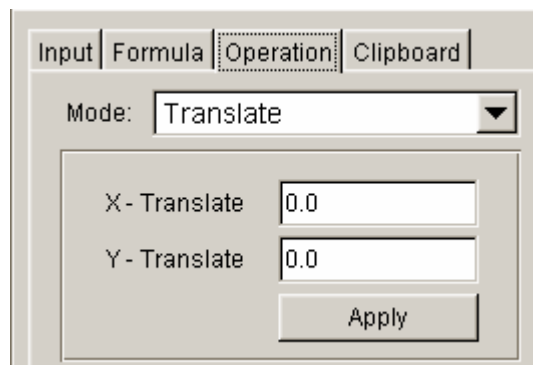


Figure 8.2.68 Curve Operation

**MODE:** Define the curve operation mode. Including SCALE and TRANSLATE.

**PARAMETER:** The user may input curve operation parameters in the input boxes. For example, during curve scale, the user needs to individually enter the scale factors for X-axis and Y-axis. During curve translation, the user needs to enter translate value for X-axis and Y-axis.

**APPLY:** After specifying operation parameters, the user may click APPLY to operate. After operation, the resulting curve is displayed automatically in the curve display window.

## CLIPBOARD

This function allows the user to temporarily save the current curve to the clipboard of Auto Setup. The user may load the curve saved in the clipboard into the current curve interface. At the same time, comparison between the curve in clipboard and the current curve is also available. Figure 8.2.69 illustrates the clipboard interface.

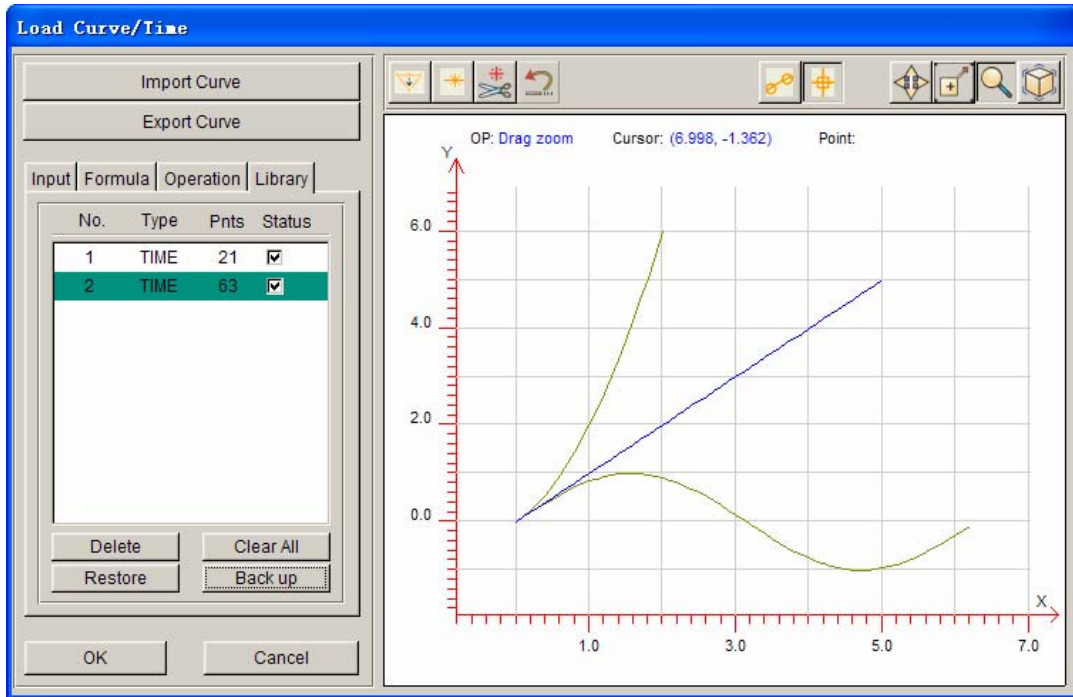


Figure 8.2.69 Clipboard Interface

**DELETE:** The user may select and delete curves from curve clipboard.

**CLEAR ALL:** The user can clear all data stored in clipboard.

**RESTORE:** Allow the user to select curves from the clipboard and restore as the current curve. If the user has already defined current curve, the program will prompt the user to use the curve in clipboard to overwrite the current curve. The program displays a dialog box, as shown in Figure 8.5.70 to prompt the user for confirmation.

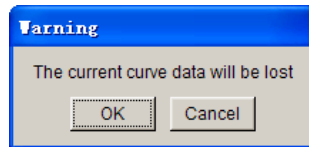


Figure 8.5.70 User Prompt dialog box

**BACK UP:** Copy the current curve to the curve clipboard. The program automatically adds a copied curve in the clipboard. All curve information in the clipboard is displayed.

NO.: Number of curves in clipboard.

TYPE: Curve type. Common curve type is time curve.

PNTS: Number of points on the curve.

STATUS: The user may control whether the curve in clipboard is displayed in show curve window. If the option is selected, the curve is displayed in gray color in the show curve window (current curve in blue color).

*Note: The curve data on clipboard is not saved in database file. After the user closed*

*eta/DYNAFORM, all the curves in clipboard are removed.*

**TOOL BAR IN SHOW CURVE INTERFACE**

The tool bar is at the top part of show curve interface. The user may operate the curves by using the functions provided in the tool bar. Tool bar interface is shown in Figure 8.5.71.

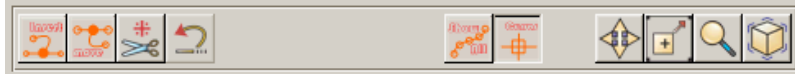



Figure 8.2.71 Curve Tool Bar

**MOVE POINT**  This function allows the user to move a point on the curve. After clicking this button, the user may move the mouse in show curve window. The program labels the point that is the closest to the cursor with a small circle. The user clicks the left mouse button to select the point. The circle turns red. The user may click and drag the mouse to move the point to a desired location. Release the left mouse button to keep the point at the new location. See figure 8.2.72 for a typical move point operation.

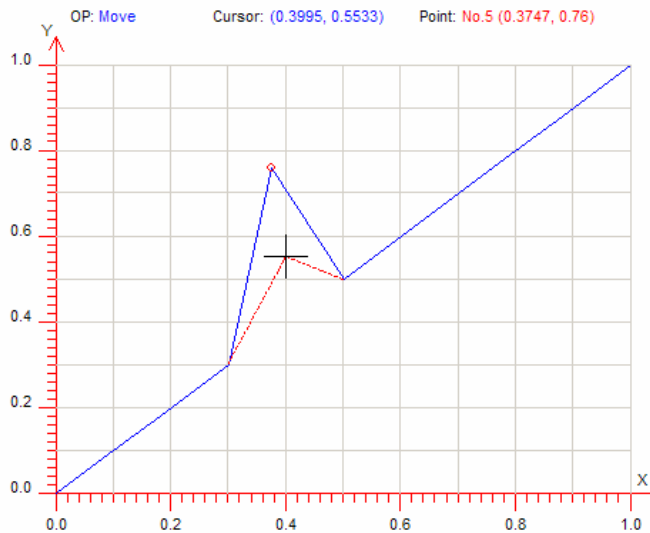





Figure 8.2.72 Move Points on the Curve


**INSERT POINT**  This function allows the user to insert a point on the curve. After clicking this button, the user may move cursor to the desired location of the curve in show curve window and click the left mouse button to insert a point.


**DELETE POINT**  This function allows the user to delete points on a curve. After clicking this button, the user may move the cursor to a point on the curve and click the left mouse button to delete the point from the curve.


**UNDO**  Allow the user to undo the previous operation.


**SHOW ALL POINTS**  This function displays a small circle on each point on the curve.




**SHOW CURSOR LINE**  This function allows the user to show cursor line.

**PAN**  This option allows user to pan the curve in the show curve window.

**ZOOM BY WINDOW**  This function allows the user to zoom in the curve by window. After clicking this function, the user needs to click and drag the left mouse button to define a rectangle region in the show curve window. Release the mouse button to zoomed in the curve.

**ZOOM BY DRAG**  This function allows the user to zoom the curve view by dragging the mouse. The user clicks left mouse button of in the show curve window. Holds the mouse button and moves the mouse downward to zoomed out. Move the mouse upward to zoom in the curve.

**FIT VIEW**  This function allows the user to fill the entire curve in the show curve window.

#### 8.2.6.6 DURATION

This function is used to control the duration of the current stage. Its interface is shown in Figure 8.2.73. The user may select four methods to define current stage's duration: direct time control, displacement control, to closure control and until home. Each of these four methods has its own advantages for different forming stage and different requirement. The user may select the appropriate control method according to the requirement of the current setup.

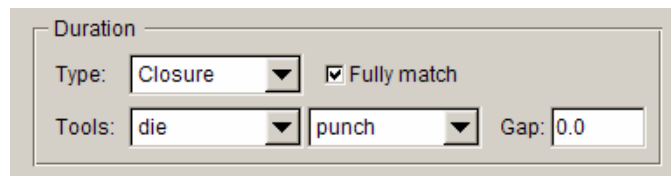


Figure 8.2.73 Stage Duration Time Define

#### DURATION TYPE

The user may select a control type from the TYPE pulling-down menu.

**TIME:** Allow the user to define a time to control the current stage calculation time. This method is usually used in hydroforming simulation. The user may enter a time value in the input box.

**TRAVEL** Allow the user to specify the current stage's calculation time by specifying the travel distance along the working direction of the tool. The user needs to select a reference tool from tool list as shown in Figure 8.2.74. Then, define the displacement value of the tool along its working direction.

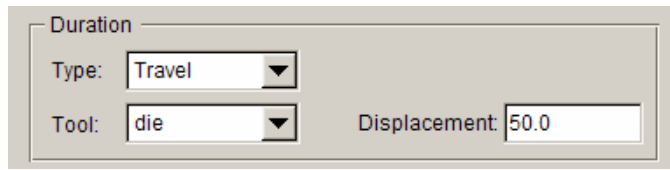


Figure 8.2.74 Displacement Control Stage Time

**NOTE:** As the working direction can be negative or positive, it is possible to set a negative value to indicate the stage time is controlled by the tool travel distance along the opposite direction of its working direction. In Auto Setup, many functions require assigned working direction. Therefore, the working direction definition is very important.

**CLOSURE** Allow the user to use the closure time of two tools as the duration time. The user needs to select two reference tools from tool list as shown in Figure 8.2.68.

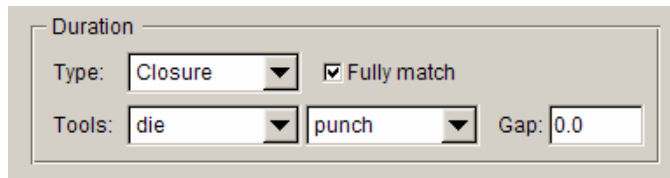


Figure 8.2.75 Displacement Control Stage Time

**FULLY MATCH:** This option should be selected when the mesh of two selected tools don't have physical offset. It is not required for the tools with physical offset.

**GAP:** Allow the user to specify the distance from complete closure of the two selected tools to stop the stage calculation. As the calculation time is based on the mesh of the two tools, the thickness of blank has to be taken into account. If the user selects physical offset meshes to calculate the closure time it is required to define the blank thickness as the gap.

**NOTE:** When using two tools' closure option, one of the tools has to be stationary, while the other must be controlled by velocity/displacement.

**UNTIL HOME** Facilitate the movement of selected tool to home position. The user needs to select a moving tool from the tool list as shown in Figure 8.2.76.

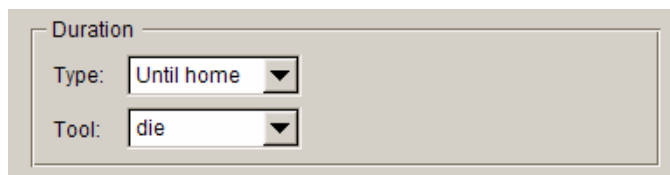


Figure 8.2.76 Until home

### 8.2.6.7 OUTPUT D3PLOT FILE CONTROL

This function allows the user to define the number of output steps in D3PLOT files. Under default condition, D3PLOT files are output in equal time interval. In advanced parameters, the user may enter additional time points or specify tool travel distance to insert the required frames at the specific time.

#### FRAMES

Define number of D3PLOT result files for the current stage calculation. The user may change this value in the input box. If specific output time is not defined, D3PLOT files will be generated with equal time interval.

#### ADVANCED

The user may insert some specified frames in advanced options. The program displays a define D3PLOT curve dialog box as shown in Figure 8.2.77.

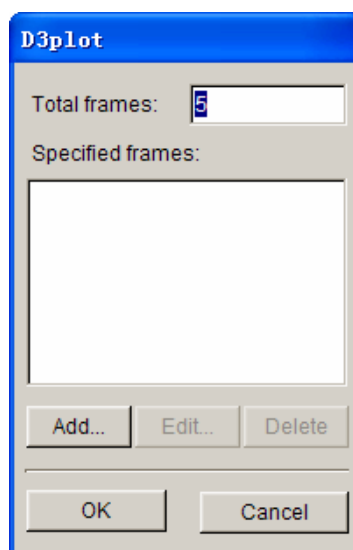


Figure 8.2.77 Add Special Output Frame

**TOTAL FRAMES** Display the total number of output frames in the D3PLOT files for the current stage.

**ADD...** The user may click this button to add special output frames. The program displays the specified frame dialog box as shown in Figure 8.2.78.

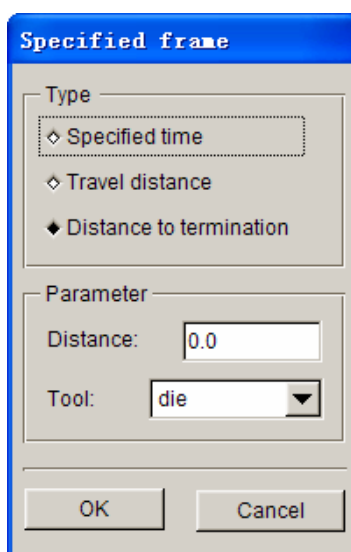


Figure 8.2.78 Specified Frame Define

**SPECIFIED TIME:** Add an output frame by specified time. See Figure 8.2.79 for the interface of this option. The user may enter specified time value in the TIME input box at the lower part of the dialog box. Note: Specified time 0 is the stages starting point. The user may add more time points to control D3PLOT file output at the specified time. If the user enters longer time than current stage calculation time, the program will ignore the invalid input value. Click OK button to complete the operation.

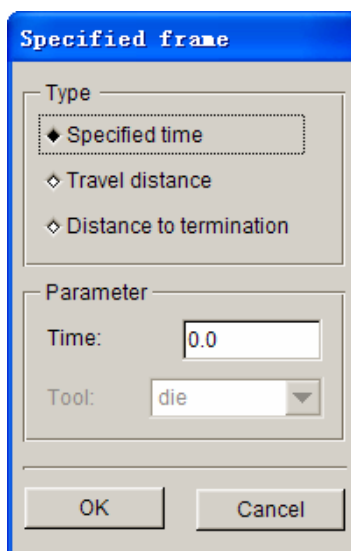


Figure 8.2.79 Input Specified Frame for Specified Time

**TRAVEL DISTANCE:** Add an output frame by specifying travel distance of the tool. See Figure 8.2.80 for the interface of this option. The user may specify a reference tool in tool pulling-down list, then enter a distance value in the DISTANCE input box. This distance value is the distance of the tool travel from the beginning of current stage.

**NOTE:** *The selected reference tool must be controlled by velocity in the current stage. Otherwise, the specified distance is meaningless.*

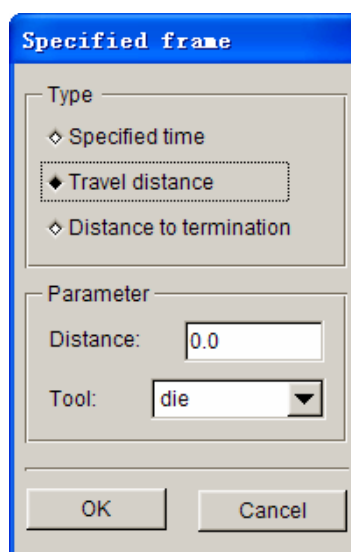


Figure 8.2.80 Define Output Frame by Tool Travel Distance

**DISTANCE TO TERMINATION:** Add an output frame by defining the distance to the stage termination. See Figure 8.2.81 for the interface of this option. The user may select a reference tool in TOOL pulling-down list, and then enter a distance value in the DISTANCE input box. This distance value is the distance of the tool from calculation termination.

**Note:** *The selected reference tool must be controlled by VELOCITY. Otherwise, the specified distance is meaningless.*

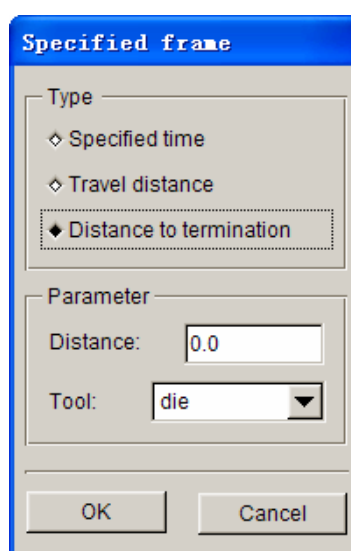


Figure 8.2.81 Define Specified Frame by Distance to Termination

After clicking OK, the added specified frame is listed in specified frame list as shown in Figure 8.2.82. Each item in the list shows the control type of the frame, tool name and it's travel distance as

appropriate.

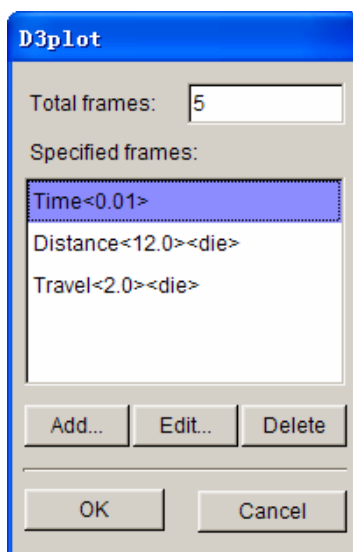


Figure 8.2.82 Specified Frame List

**EDIT...:** The user may select a specified frame from the specified frame list to edit.

**DELETE:** The user may delete a specified frame from the specified frame list.

**OK:** Save current setup result and exit the specified frame setup dialog box.

**CANCEL:** Exit setup dialog box without saving current setup.

After the user added specified frames, the program shows the number of specified frames in the current stage. An example is illustrated in Figure 8.2.83.

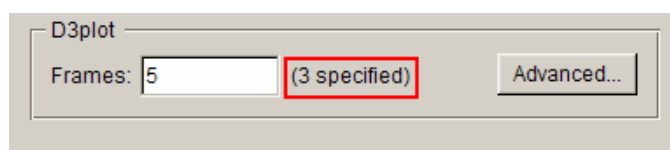


Figure 8.2.83 D3PLOT File of Special Output Time Point

### 8.2.6.8 HYDRO MECHANICAL

This function allows the user to set parameters for hydro mechanical forming . The user may select Hydro mech. Option as shown in Figure 8.2.84. The user may select the direction of applied fluid pressure on blank.

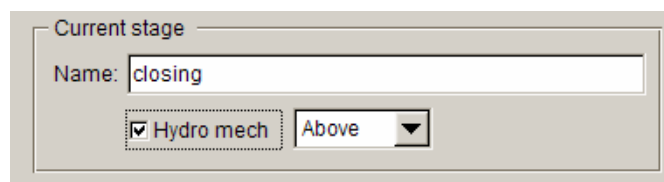


Figure 8.2.84 Hydro Forming

**ABOVE:** In press coordinate system, fluid pressure is applied on the top face of blank as shown in Figure 8.2.85.

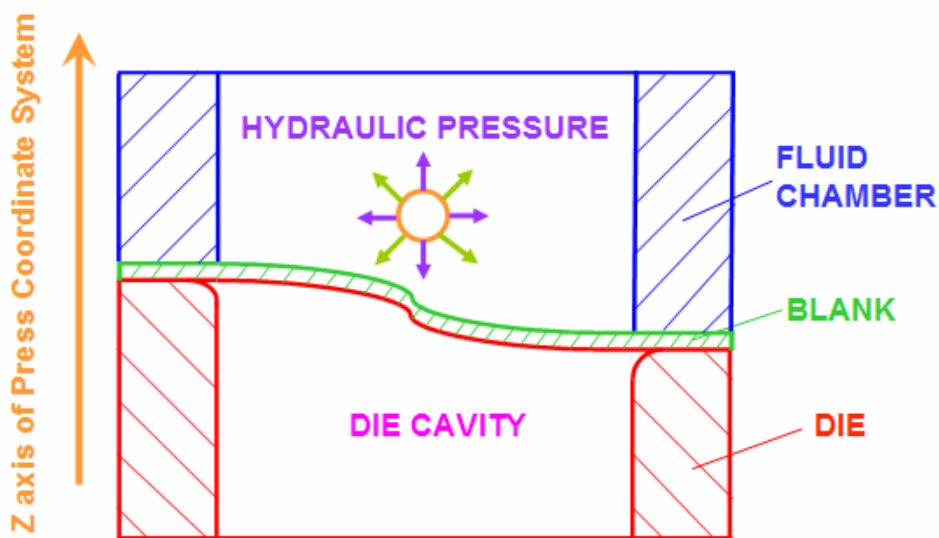


Figure 8.2.85 Fluid Pressure Applied on Top Face of Blank

**BELOW:** In press coordinate system, fluid pressure is applied on the bottom face of blank as shown in Figure 8.2.86.

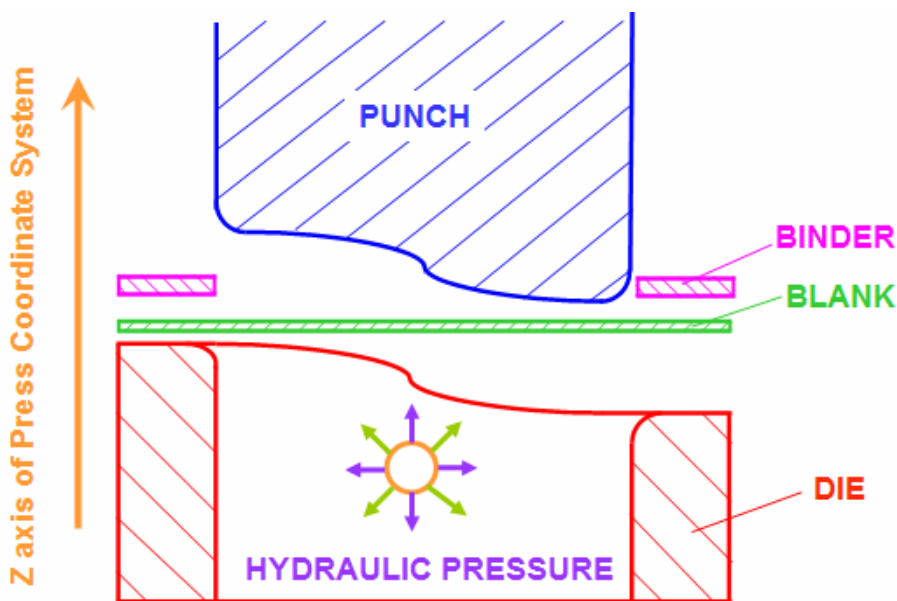


Figure 8.2.86 Fluid Pressure Applied on bottom face of blank

**BY NORMAL:** Direction of fluid pressure is defined according to blank element normal. The user needs to check the blank element normal carefully, and make sure the element normal are consistent. Otherwise, the analysis will not be correct.

### HYDRO MECH. PARAMETERS

In hydro mech. Parameter setup, the user may specify hydro pressure-time curve and the scope of the pressure. The interface is shown in Figure 8.2.87

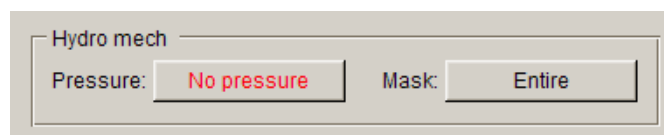



Figure 8.2.87 Hydro Mech. Parameters Setup

**PRESSURE:** Set up hydro pressure-time curve. This parameter is essential in sheet hydro forming.

The user may click  button to enter hydro pressure- time curve in the define dialog box as shown in Figure 8.2.88.

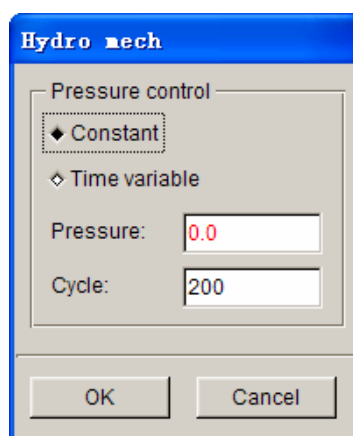


Figure 8.2.88 Hydro Pressure-Time Curve Defines

**CONSTANT:** Define hydraulic pressure that does not change with time. The user may enter a constant fluid pressure in Pressure input data field.

**CYCLE:** Define analysis cycle time of fluid pressure application. Default is to renew for every 200 calculation cycles.

**TIME VARIABLE:** Define variable hydraulic pressure curve with time. The user may click EDIT button to enter curve define interface. For detailed description of curve definition, refer to Chapter 12.

**MASK:** Assign mask of applied fluid pressure using closed loop cutting line. Applied fluid pressure can be either inside of the mask, or outside. It is not necessary to define this parameter. The program by default applies fluid pressure on all blank elements. Click the Entire button to display the MASK dialog



box as shown in Figure 8.2.89.

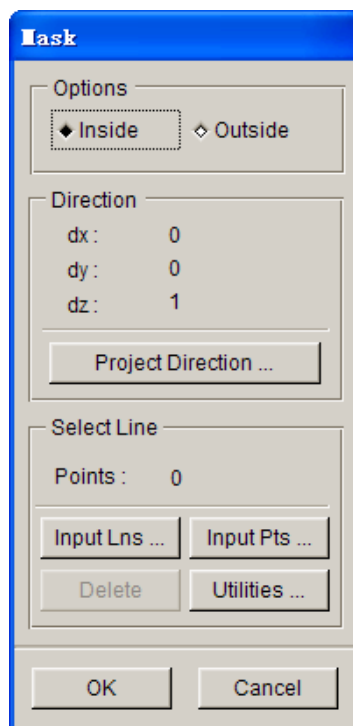


Figure 8.2.89 Define Fluid Pressure Apply Cutting line

**INSIDE** The user may define fluid pressure inside the defined cutting line.

**OUTSIDE** The user may define fluid pressure outside the defined cutting line.

**PROJECT DIRECTION...:** Allow the user to select curve project direction. eta/DYNAFORM adopts one closed loop curve for the definition of fluid pressure mask, hence it is not necessary for the curve to be positioned on top of blank. Only the assigned curve is needed for projection onto the blank. Therefore, the project direction for closed curve is required. After the user click this button, the direction define dialog box is displayed. Detailed description of this function is provided in Section 8.2.3.2. The default project direction is Z-axis of the working coordinate system.

**DX DY DZ:** Display the coordinates of the end point of project direction vector. Default vector starting point is the origin of WCS.

**POINTS:** Display the number of points on curve.

**INPUT LNS...:** The user may define region subjected to applied fluid pressure by defining a closed loop curve as cutting line. After clicking this button, the SELECT CURVE dialog box is displayed.

**INPUT PTS...:** If no curve is provided in the database, the user may define a closed loop curve by clicking this button to create the cutting line using points.

**Notice:** When defining mask to applied fluid pressure, it is better to select the closed loop curve or a closed end point. If the curve/point is not closed, the program automatically closes curve and point.

**DELETE:** The user may delete current defined cutting line. The program displays a confirmation dialog box as shown in Figure 8.2.90.

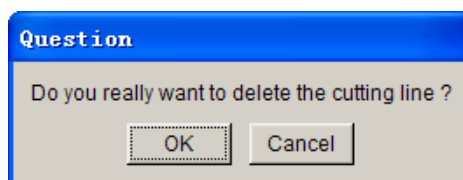


Figure 8.2.90 Confirmation dialog box

**UTILITIES...:** The user may click this button to view curve functions. The user may create the cutting line in curve define dialog box.

**OK:** Accept the current setup and exits the dialog box.

**CANCEL:** Discard the current setup and exits the dialog box.

If the user has already defined the region subjected to applied fluid pressure, the PARTIAL is displayed on the button next to MASK. This indicates only part of elements on the blank is subjected to applied fluid pressure.

### 8.2.6.9 GRAVITY

GRAVITY stage is used to simulate the deformation of blank under gravity load. The settings are similar to normal sheet forming process simulation. It includes basic parameters, blank define, tool define, etc. The interface of GRAVITY stage is shown in Figure 8.2.91.

**NOTE:** Gravity analysis cannot exist in the same setup with other type of analysis.

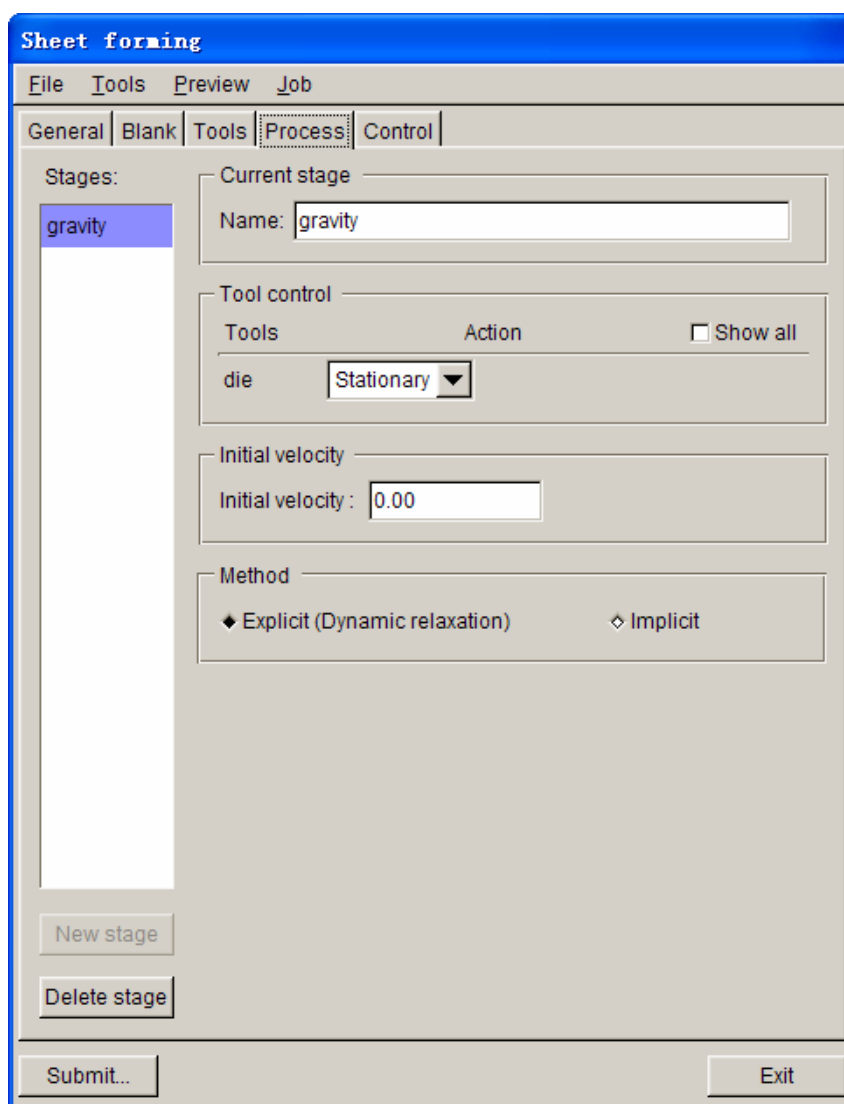


Figure 8.2.91 Gravity Analysis Process Parameters

**INITIAL VELOCITY:** Allow the user to specify initial velocity of the blank.

**METHOD:** Allow the user to select gravity analysis method: explicit (dynamic relaxation) or implicit. The difference between the characteristics of these two methods is described in LS-DYNA User's Manual.

#### 8.2.6.10 SPRINGBACK

The SPRINGBACK under process template is used to simulate the springback of the blank after draw simulation. See Figure 8.2.92.

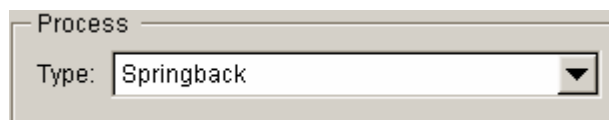


Figure 8.2.92 Springback Analysis

**NOTE:** Springback analysis cannot exist in the same setup with other type of analysis.

Springback analysis process parameters setup is shown in Figure 8.2.93.

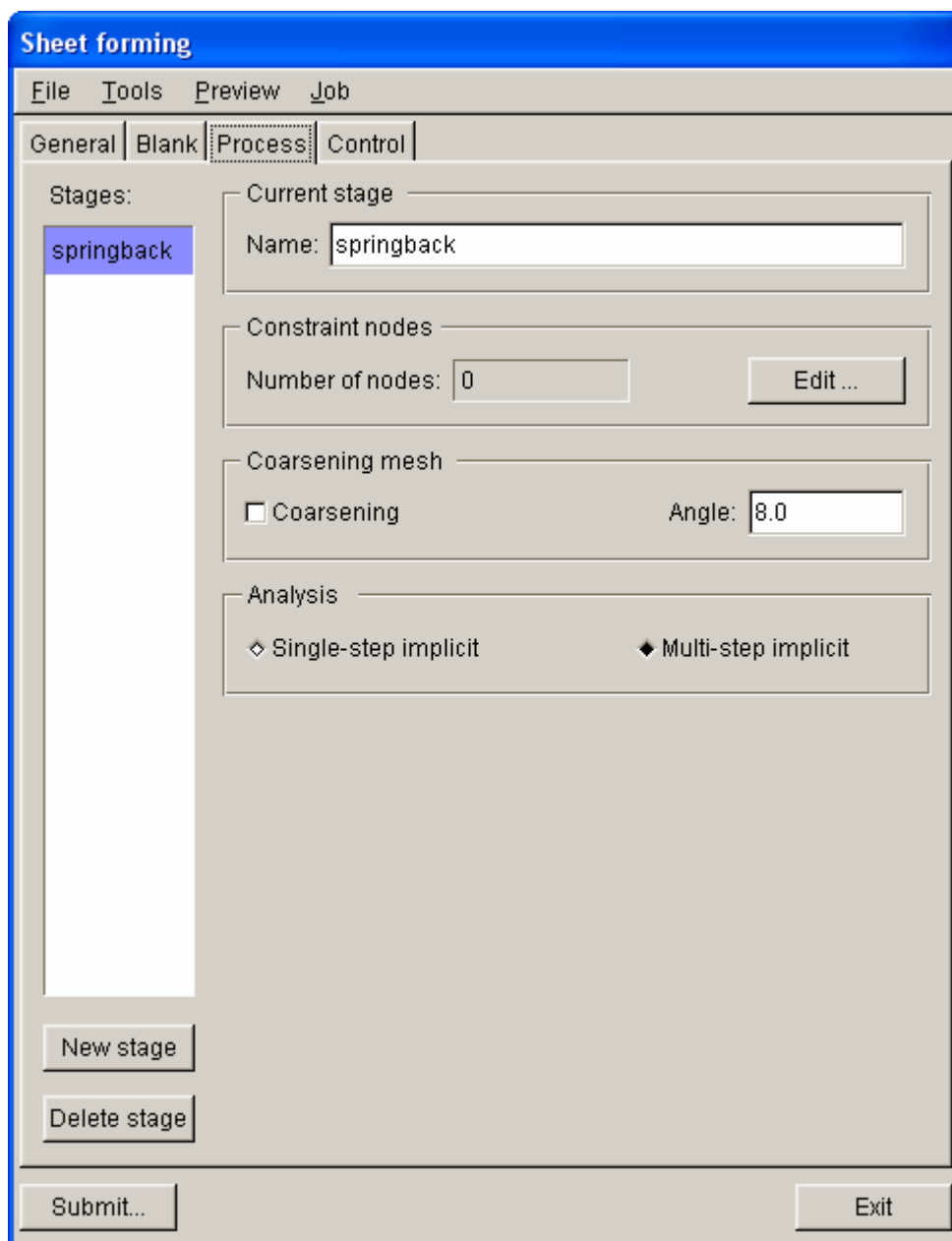


Figure 8.2.93 Springback Analysis Process Parameters

### CONSTRAINT NODES

Allow the user to select three nodes to constrain the rigid displacement of the blank. The user is prompted to select nodes in NUMBER OF NODES dialog box. The user may click the EDIT button to select three nodes. The program displays the select node dialog box as shown in Figure 8.2.94. The

user may select three constraint nodes on the screen. The constrained degree of freedom of the selected constraint nodes is displayed on the screen.

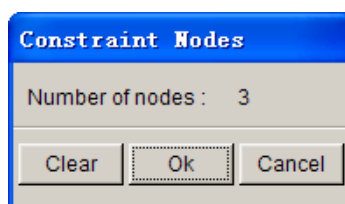


Figure 8.2.94 Constraint Node Define

**NOTE:** *The first selected node is constrained in X, Y, Z direction. The second selected node is constrained in Y, Z direction. The third selected node is constrained in Z direction.*

**CLEAR:** Clear all defined constraint nodes. The user may re-define the constraint nodes.

**OK:** Save current setup and exits the dialog box.

**CANCEL:** Exit the dialog box without saving current change.

## COARSENING MESH

Allow the user to use coarsening mesh technique during springback calculation. During drawing calculation, adaptive mesh technique is adopted to refine the blank mesh. It results in a large number of elements for springback calculation, which brings great challenge to the convergence in implicit solution. This option will merge the adapted mesh in the flat areas prior to springback calculation. It will reduce convergence problem and shorten the computing time.

**ANGLE:** Set the angle criteria for merge neighboring elements. When the element normal angle difference between adjacent elements is less than the given criteria, the program merges these elements. Default angle is 8 degrees.

## ANALYSIS

Allow to user to select springback analysis calculation method: single action springback and multiple actions springback calculation. Single action springback is the default option. Single action springback calculation usually leads to good result. For relatively soft materials, convergent result may not be obtained in single action calculation. The user may consider to use multiple actions springback calculation. Detailed descriptions for these calculation options can be found in LS-DYNA User's Manual.

## 8.2.7 CONTROL

In Control parameter page, the user is allowed to set up basic parameters to control calculation. It includes step size setup, specify number of CPUs, blank mesh adaptive setup, implicit analysis parameters setup and restart file output control, etc. Control parameter setup interface is shown in Figure 8.2.95.

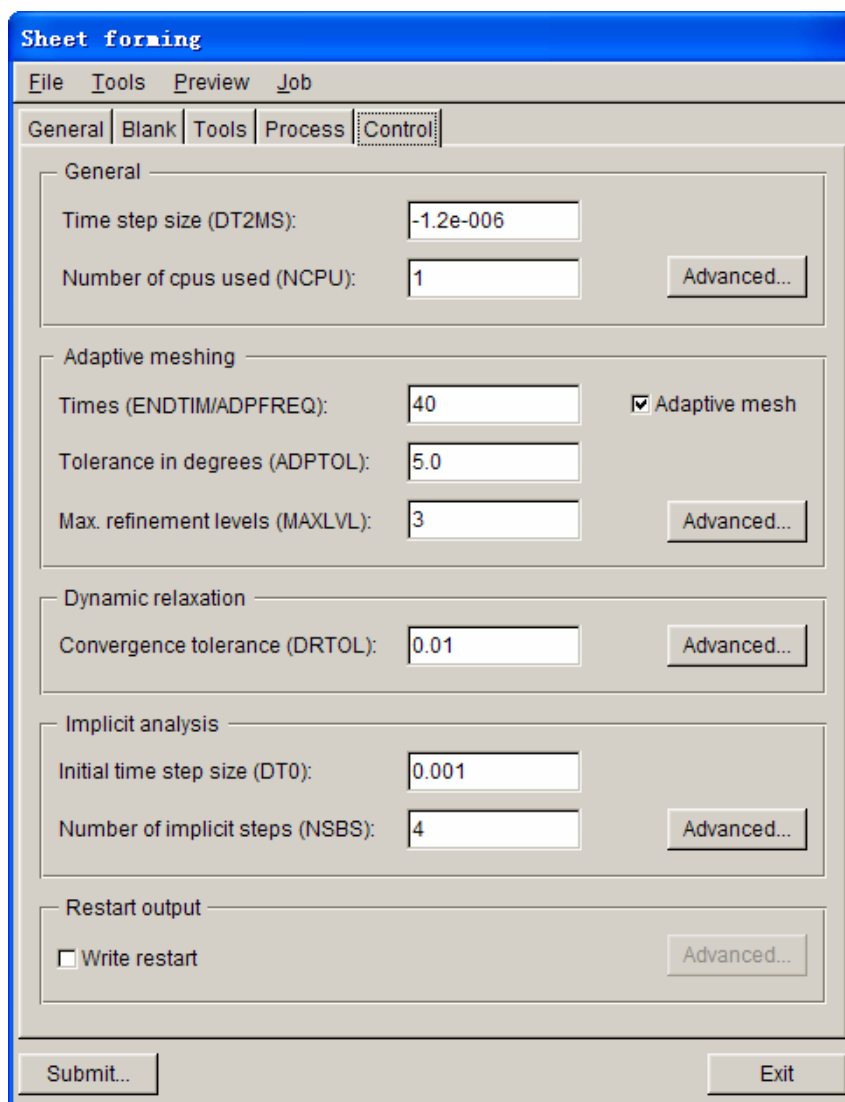


Figure 8.2.95 Basic Control Parameter Setup

### 8.2.7.1 GENERAL

**TIME STEP SIZE:** Allow the user to set up time step size. Default time step size is -1.2e-6.

**NUMBER OF CPUs USED (NCPU):** Define number of CPUs used in calculation. Default value is 1.

**ADVANCED...:** Advanced setup for basic parameters. The interface is shown in Figure 8.2.96.

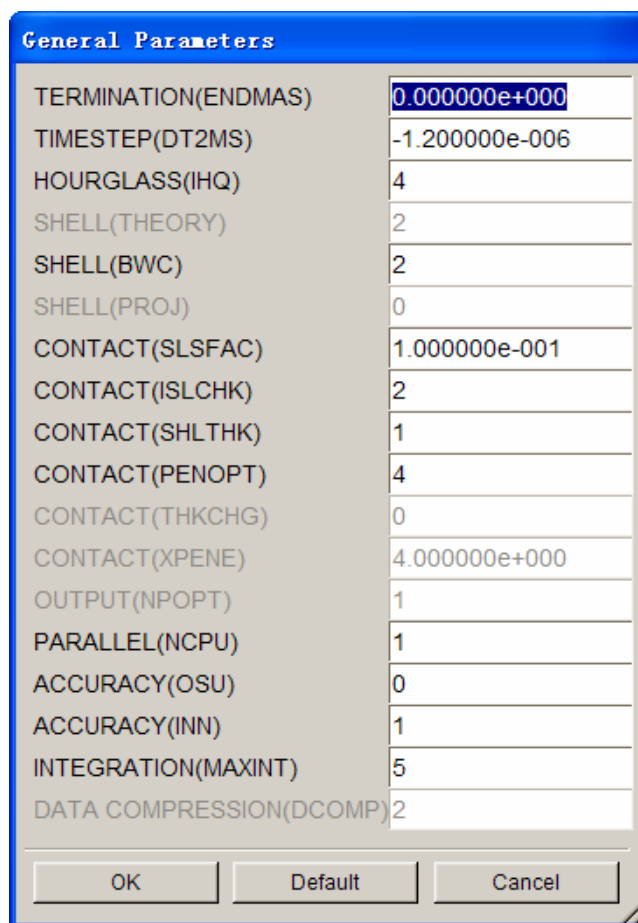


Figure 8.2.96 Basic Parameter Advanced Setup Interface

### 8.2.7.2 ADAPTIVE MESHING

**TIMES (ENDTIME/ADPFREQ):** Allow the user to set up the adaptive check time interval. Default time separation is to check every 40 cycles.

**TOLERANCE IN DEGREE (ADPTOL):** Set up angle change between elements during adaptive meshing. If angle change is greater than 5 degrees, the coarse elements are divided into fine elements.

**MAX. REFINEMENT LEVELS (MAXLVL):** Maximum refinement level allowed. Default level is 3. It means one element can mostly be divided into 4 elements after adaptive meshing.

**ADVANCED...:** Advanced setup for adaptive meshing. The interface is shown in Figure 8.2.97.

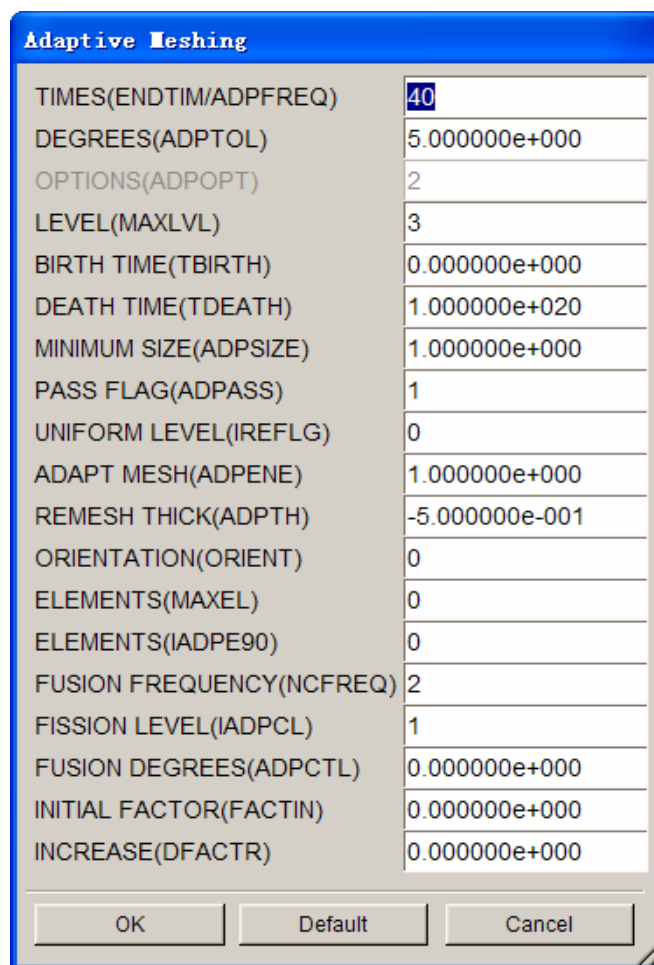


Figure 8.2.97 Adaptive Advanced Parameters

### 8.2.7.3 DYNAMIC RELAXATION

**CONVERGENCE TOLERANCE (DRTOL):** Setup convergence tolerance during dynamic relaxation calculation. Default convergence tolerance is 0.01.

**ADVANCED...:** Advanced parameter setup of dynamic relaxation. The interface is shown in Figure 8.2.98.



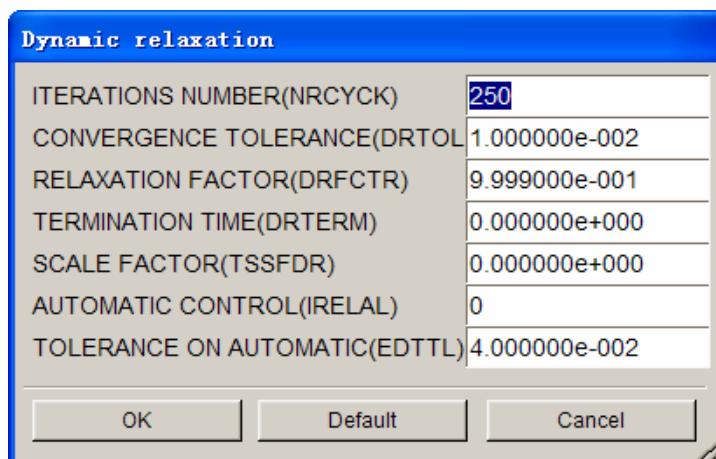


Figure 8.2.98 Advanced Parameters of Dynamic Relaxation Calculation

#### 8.2.7.4 IMPLICIT ANALYSIS

**INITIAL TIME STEP SIZE (DT0):** Initial time step size of springback analysis. Default value is 0.001.

**NUMBER OF IMPLICIT STEP (NSBS):** Number of implicit step. If the value is 1, it indicates single step implicit analysis. If the value is greater than 1, multiple implicit analysis will be used.

**ADVANCED...:** Advanced parameter setup of implicit analysis. See Figure 8.2.99 for the list of parameters.

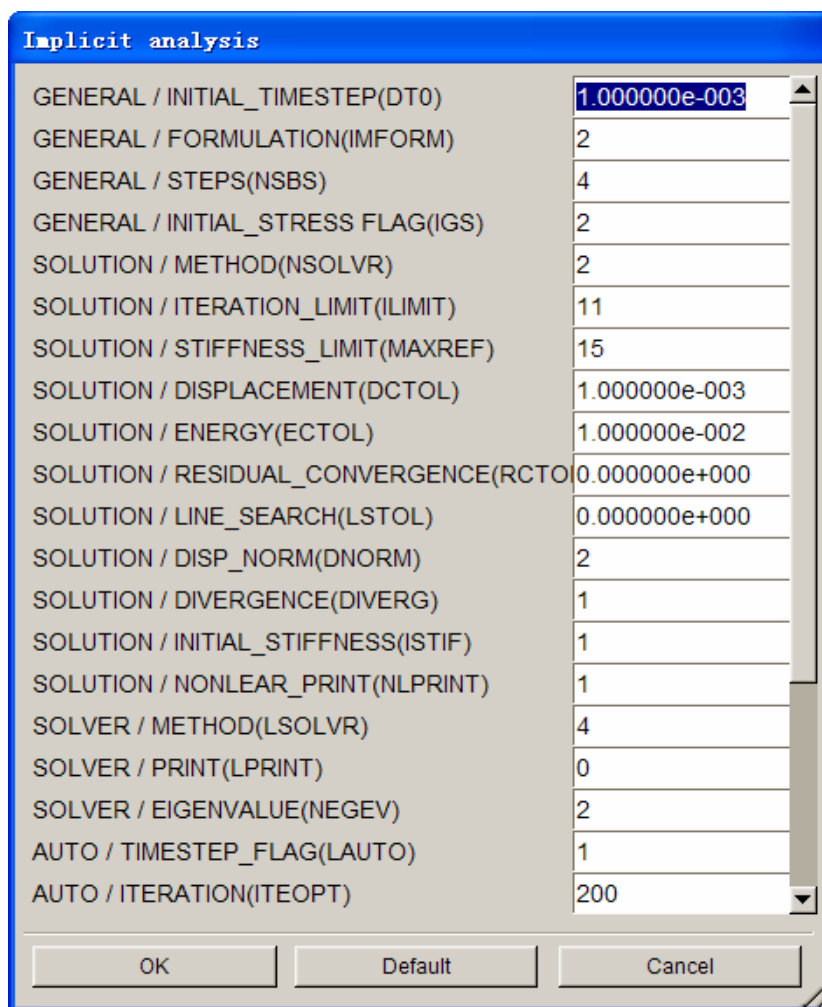


Figure 8.2.99 Advanced Parameter Setup of Implicit Analysis

### 8.2.7.5 RESTART

**WRITE RESTART:** The user may use this option to output one or several restart files. The interface is shown in Figure 8.2.100.

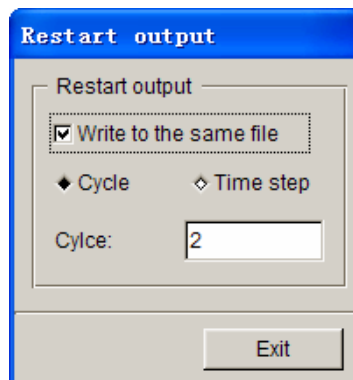


Figure 8.2.100 Restart Advanced Parameter Setup

Detailed description of restart can be found in LS-DYNA User's Manual.

## 8.2.8 TUBULAR HYDROFORMING

The process of tubular hydro forming is basically the same as normal sheet forming. The user may refer to the setting of sheet forming described in previous sections. Only specialized setup options related to tubular hydroforming are provided here.

### 8.2.8.1 NEW SIMULATION

The user may create a new simulation, and select TUBULAR HYDROFORMING from simulation type drop-down list, as shown in Figure 8.2.101.

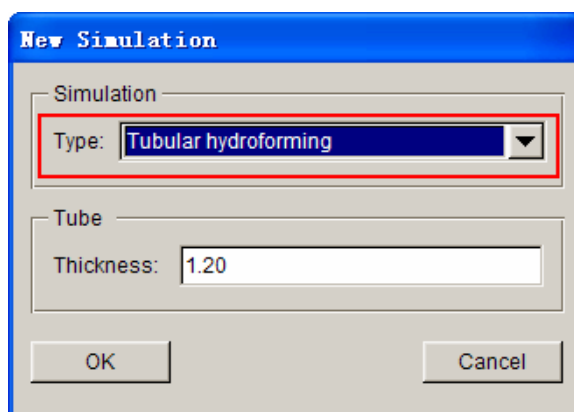


Figure 8.2.101 Tubular Hydro Forming

The user may enter the blank initial thickness. Click OK button to enter the setup interface.

### 8.2.8.2 GENERAL

Same as normal sheet forming, the user may specify simulation title, working coordination system, acceleration time and tool gap between tool and blank. The interface is shown in Figure 8.2.102. Detailed description of the GENERAL is provided in Section 8.2.3.

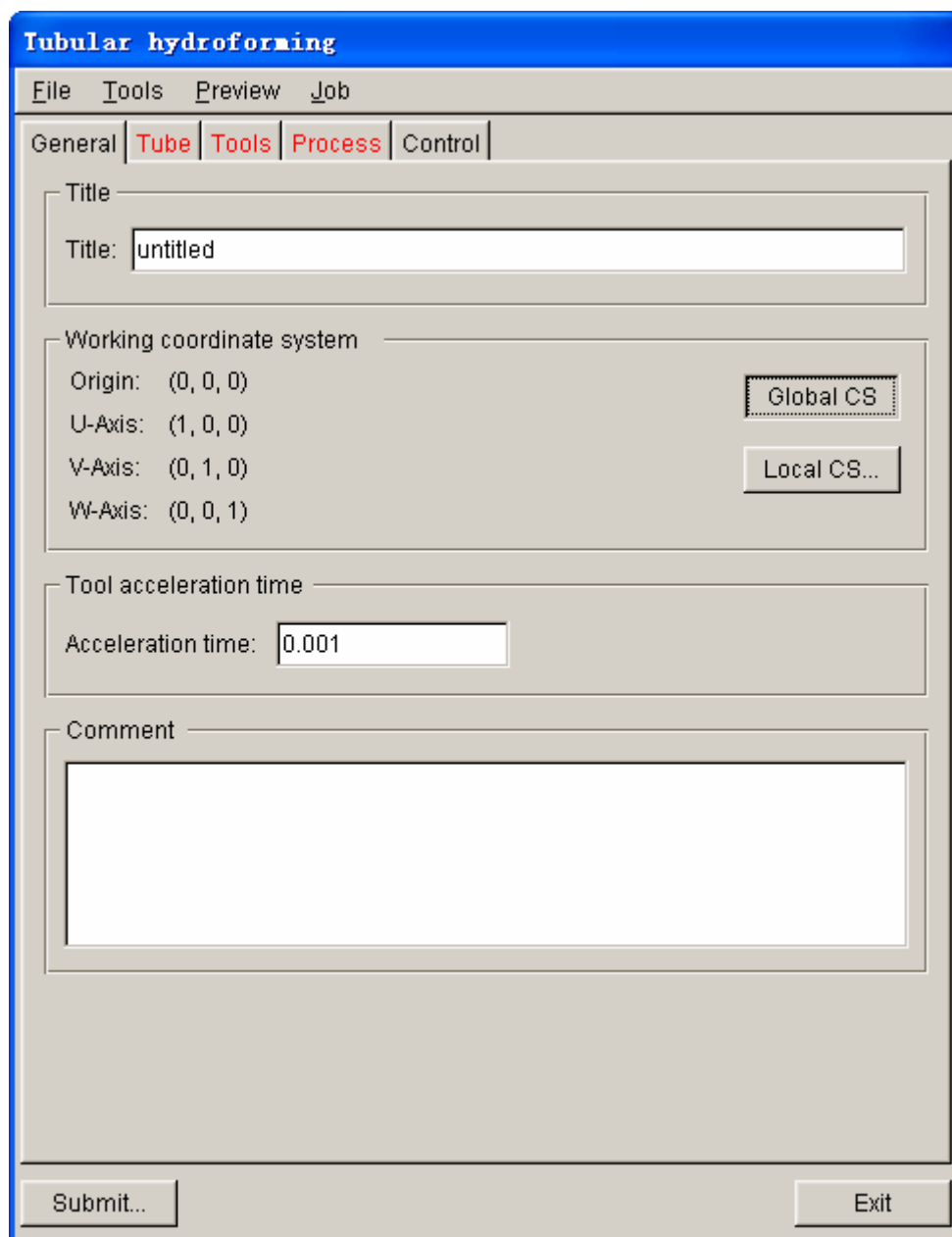


Figure 8.2.102 Tubular Hydro Forming Basic Parameter

### 8.2.8.3 TUBE

This function allows the user to specify the tube. The user may click PART button to select part for the tube. The interface is illustrated in Figure 8.2.103. For blank definition, refer to Section 8.2.4.

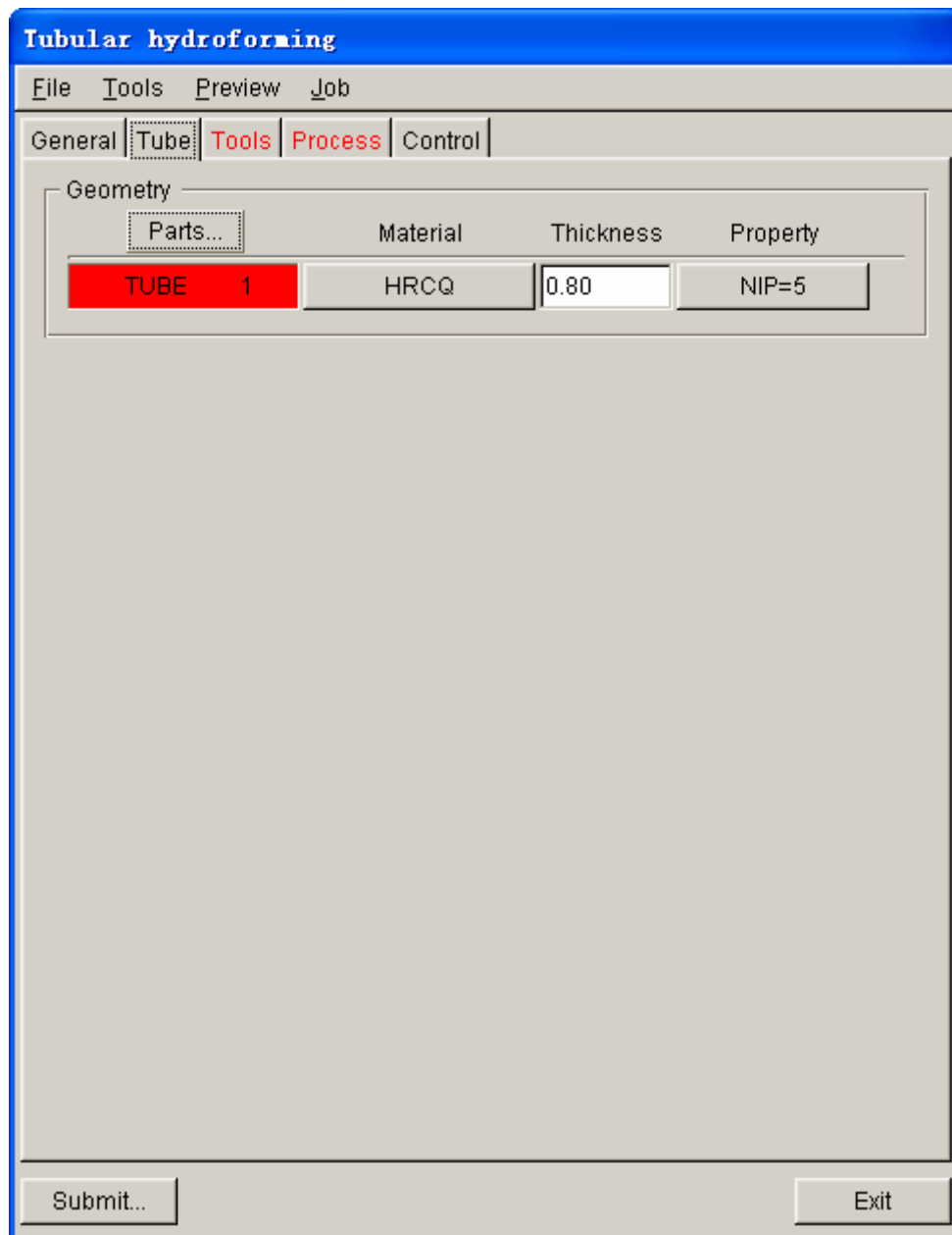


Figure 8.2.103 Part Definition of Tube

#### 8.2.8.4 TOOLS

This function allows the user to specify tools. The interface of the tool definition is illustrated in Figure 8.2.105. For description of the tool setup, refer to Section 8.2.5.

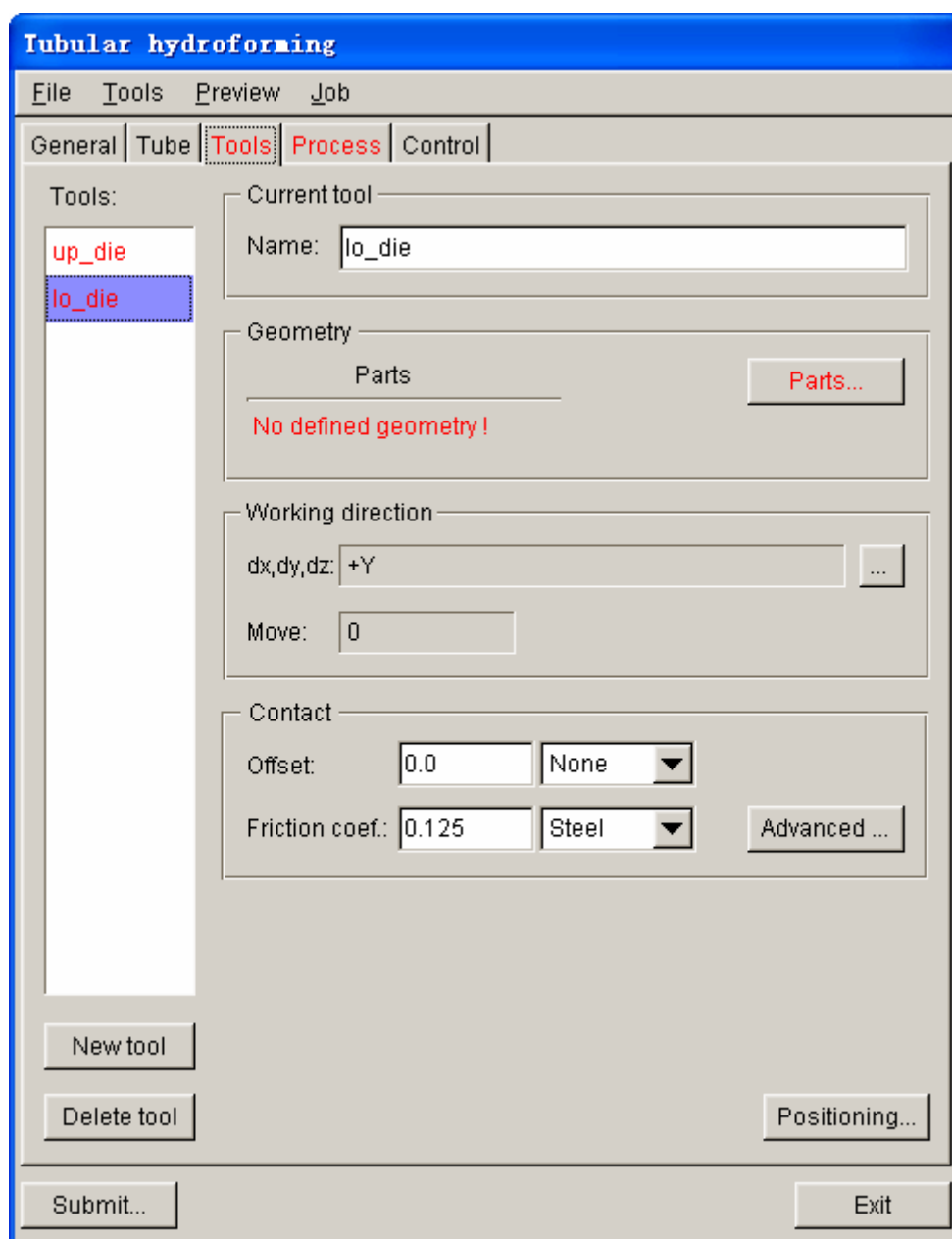


Figure 8.2.104 Tool Definition

### 8.2.8.5 PROCESS

This function allows the user to set up process parameters for tubular hydroforming. It's basic setting is same as sheet hydroforming. The user may refer to Section 8.2.6 for detailed description. For Tubular Hydroforming, the user may specify fluid flux to control hydraulic pressure. The setup interface is shown in Figure 8.2.105. The user needs to select Hydro Mechanical option, following by selecting the VOLUME type from drop-down list.

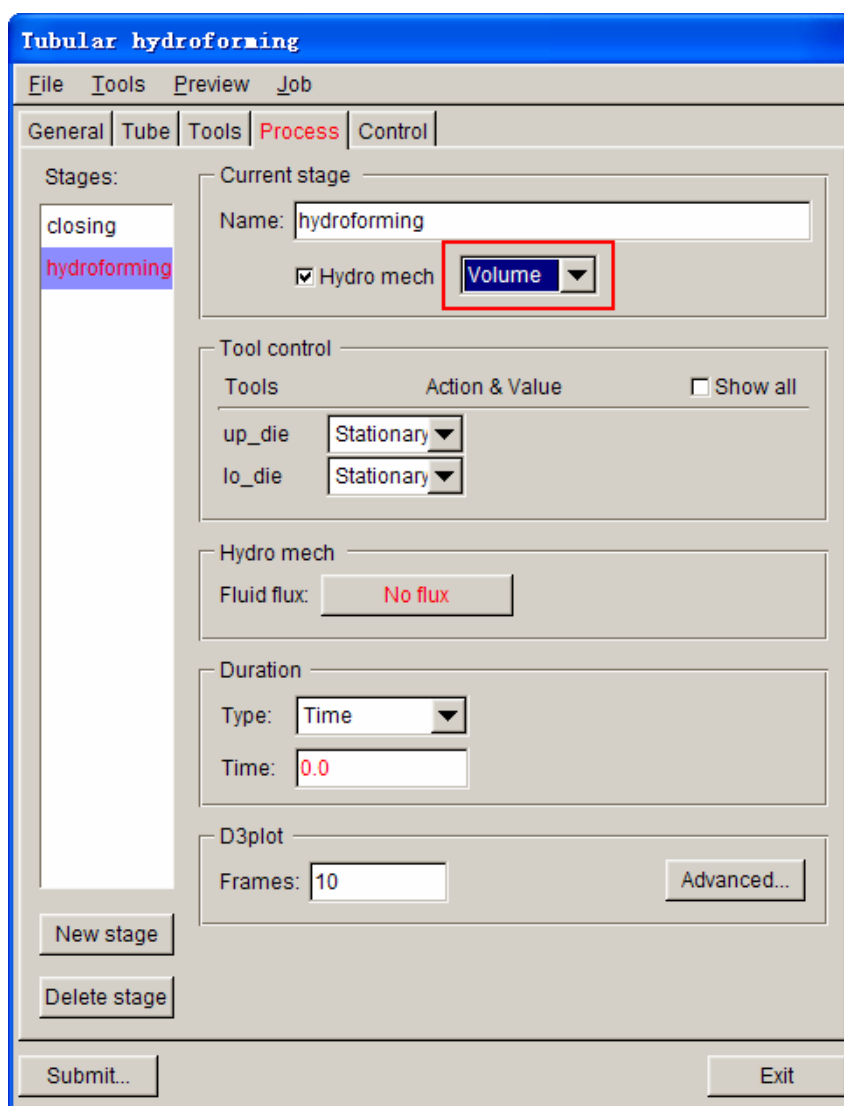


Figure 8.2.105 Hydro Control Parameters

**FLUID FLUX:** Allows the user to specify fluid flux – time curve to control fluid pressure. The program displays the fluid flux setup dialog box as shown in Figure 8.2.106.

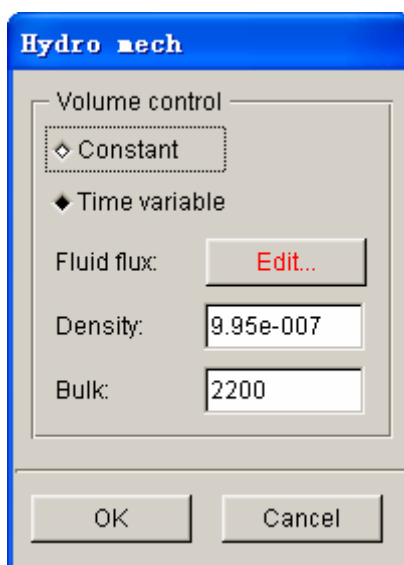


Figure 8.2.106 Fluid Flux Control dialog box

**CONSTANT:** Defines constant fluid flux. The user may enter a constant fluid flux value in FLUID FLUX input box.

**TIME VARIABLE:** Defines time variable fluid flux. The user may click EDIT button next to FLUID FLUX to enter curve definition interface. Detailed description of curve definition is provided in Chapter 12.

**FLUID FLUX:** Defines flux of fluid.

**DENSITY:** Defines density of fluid.

**BULK:** Defines bulk viscosity of fluid.



## Chapter 9

# TOOL DEFINITION

The functions provided in this menu are shown in Figure 9.1. The user can set up tool definition, define materials and properties, create the guide/pin, generate/modify load curves, define contact interface, position tools, define draw bead, view the movement of defined tools and modify deformed blank shapes.

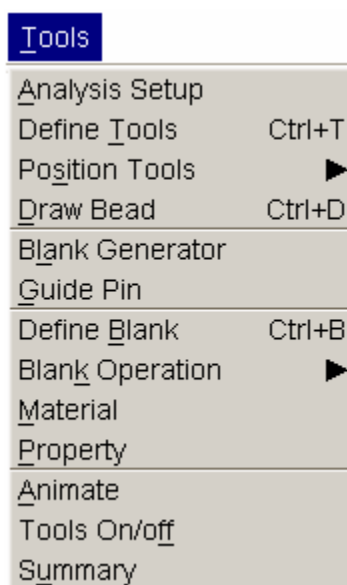


Figure 9.1 Tool Menu

A detailed description of each function and corresponding submenu is given in the following sections.

## 9.1 ANALYSIS SETUP

This function enables the user to define forming parameters for the LS-DYNA ANALYSIS. The dialog box is shown in Figure 9.1.1.

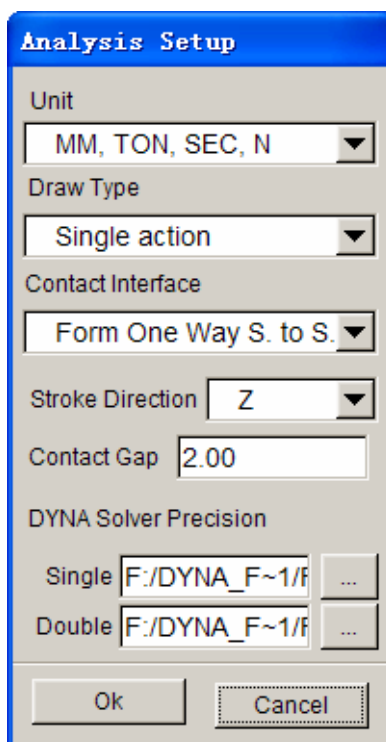


Figure 9.1.1 The Analysis Setup dialog box

### UNIT

The user can select a UNIT system by clicking the drop down button of UNIT. As shown in Figure 9.1.2, four types of UNIT systems are available. The default UNIT system is set as <MM, TON, SEC, N>.

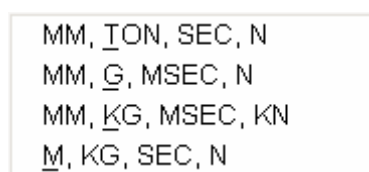


Figure 9.1.2 The Unit System Drop Down Menu

### DRAW TYPE

The user can select a draw type by clicking on the drop down button of DRAW TYPE to display the menu as illustrated in Figure 9.1.3.

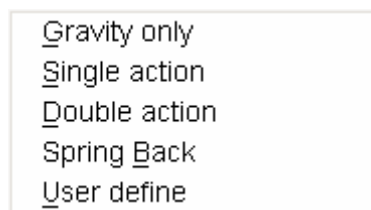


Figure 9.1.3 The Draw Types Drop Down Menu

**GRAVITY ONLY**

No punch and die is used as the blank is deformed by gravity.

**SINGLE ACTION**

The die (female piece) is above the blank.

**DOUBLE ACTION**

The die (female piece) is below the blank.

**SPRING BACK**

Relieve residual stresses of deformed sheet metal part (imported DYNAIN file).

**USER DEFINE**

The DRAW TYPE is user defined.

**NOTE:** *The defined DRAW TYPE works in conjunction with both the BLANK OPERATION/AUTO POSITION and TOOL/AUTO POSITION functions.*

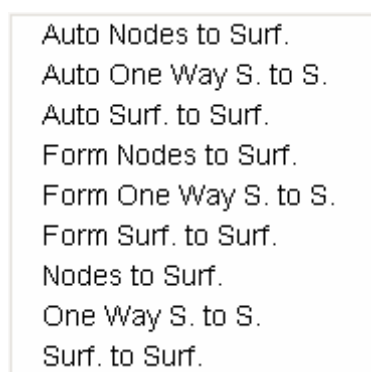


Figure9.1.4 Contact Interface Types

**CONTACT INTERFACE**

There are several types of contact interface available for an LS-DYNA stamping simulation, as listed in Figure 9.1.4. The user can set the type of contact interface between all the tooling and

the blank. The default contact interface between the blank and the tooling is set as FORM ONE WAY S. TO S.

*NOTE: Please refer to the LS-DYNA User's Manual, keyword \*CONTACT, for more detailed information about each of the contact interfaces.*


#### **STROKE DIRECTION**

The user can define stroke direction in X, Y, or Z.

#### **CONTACT GAP**

The contact gap is the minimum distance between the tooling and blank after auto-positioning. The default contact gap is set to 1.0. The contact gap will be overwritten if blank thickness is defined.

#### **DYNA SOLVER PRECISION**

This option enables user to set the directory of the Single Precision solver and Double Precision solver. User can click the  button to change the directory.

## **9.2 DEFINE TOOLS (CTRL+T)**

In eta/DYNAFORM, there are three standard tools: DIE, PUNCH and BINDER. The user can also create as many tools as necessary for any stamping simulation. The top two toggle switches in the dialog box, as shown in Figure 9.2.1, provide options for using STANDARD TOOLS setup and/or USER DEFINED TOOLS setup.

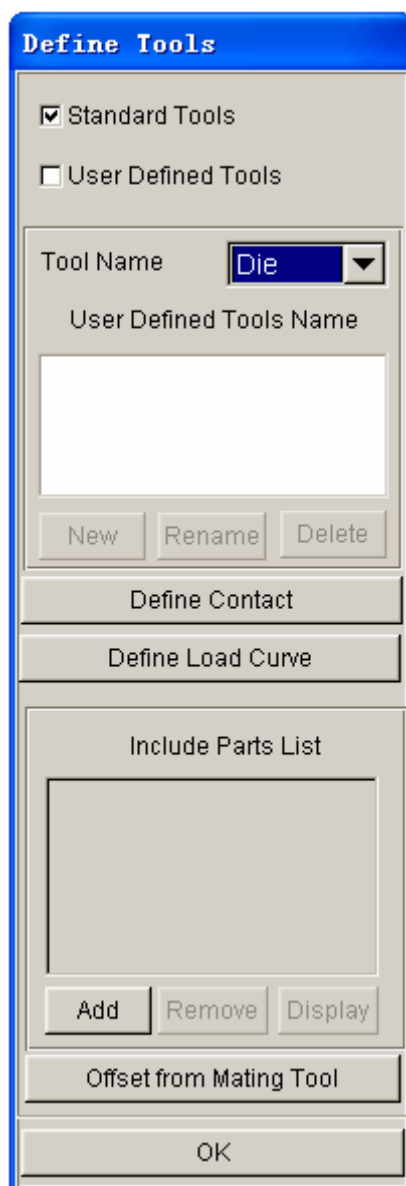


Figure 9.2.1 Define Tools dialog box

### 9.2.1 TOOLS SETUP

When checkbox of the STANDARD TOOLS is toggled on, the user can define the standard tools (DIE, PUNCH and BINDER) by selecting from the TOOL NAME drop down button as shown in Figure 9.2.1.

When checkbox of the USER DEFINED TOOLS is toggled on, the user can define and/or edit any user- defined tool by using the following functions:

**NEW**

Create a new tool name.

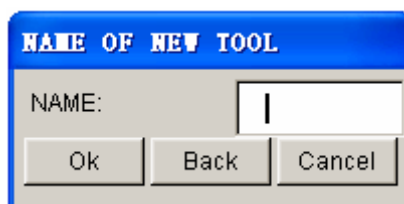


Figure 9.2.2 User Defined Tool Name Window

**RENAME**

Edit existing tool name.

**DELETE**

Remove selected tools from the USER DEFINED TOOL list.

**9.2.2 ADD (PART)**

In general, a tool consists of a set of parts. This function enables the user to add parts to the current tool. To add a part, the user needs to click on the ADD button following by choosing a part from the SELECT PART dialog box. Refer to Section 4.5 for more information. The selected parts will be listed in the INCLUDE PARTS LIST window as shown in Figure 9.2.1.

**9.2.3 REMOVE (PART)**

This function allows the user to remove parts from the current tool. To remove the part, the user needs to select/highlight a part from the INCLUDE PARTS LIST window following by clicking the REMOVE button.

**9.2.4 DISPLAY (TOOL)**

This function enables the user to show all the parts included in the current tool definition in the display area. To display the part's current tool, the user needs to click on the DISPLAY button.

**9.2.5 DEFINE CONTACT**

This function is used to define the contact interface between the blank and a tool. In most cases, the default setting, \*CONTACT\_FORMING\_ONE\_WAY\_SURFACE\_TO SURFACE, is utilized for contact between all tooling and blank. The user can modify the contact interface for individual tools.

To modify the default contact interface setting, click on the DEFINE CONTACT button as shown in Figure 9.2.1 to display the TOOLS CONTACT dialog box as shown in Figure 9.2.3.

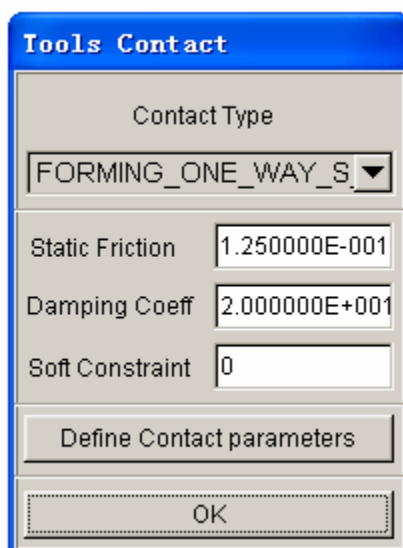


Figure 9.2.3 Define Contact dialog box

### CONTACT TYPE

Select the CONTACT TYPE by clicking on the drop down button to display the list, and select the desired type.

**NOTE:** *Nine commonly used contact interfaces in sheet metal forming are offered in eta/DYNAFORM. They are:*

1. AUTO-NODES-TO-SURF
2. AUTO-ONE-WAY-SURF-TO-SURF
3. AUTO-SURF-TO-SURF
4. FORMING-NODES-TO-SURF
5. FORMING-ONE-WAY-SURF-TO-SURF
6. FORMING-SURF-TO-SURF
7. NODES-TO-SURF
8. ONE-WAY-SURF-TO-SURF
9. SURFACE-TO-SURFACE

### DEFINE CONTACT PARAMETERS

This function allows the user to set the contact parameters such as static friction, damping coefficient, soft constraint, etc.

**NOTE:** *Refer to the LS-DYNA User's Manual for detailed information about each contact interface and its associated parameters.*

### 9.2.6 DEFINE LOAD CURVE

This function allows the user to define, read in, or modify MOTION and/or FORCE load curves. As shown in Figure 9.2.4, the user can define the curve type by toggling on either checkbox of MOTION or FORCE. The MOTION curve allows the user to define the tool movement via velocity or displacement control, while the FORCE curve allows user to define a given force applied onto a tool. If both the MOTION and FORCE curves are defined for a tool, the MOTION curve will overwrite the FORCE curve control. However, the force control can be activated if a death time is set for the MOTION curve (the default death time is the termination time of the MOTION curve).

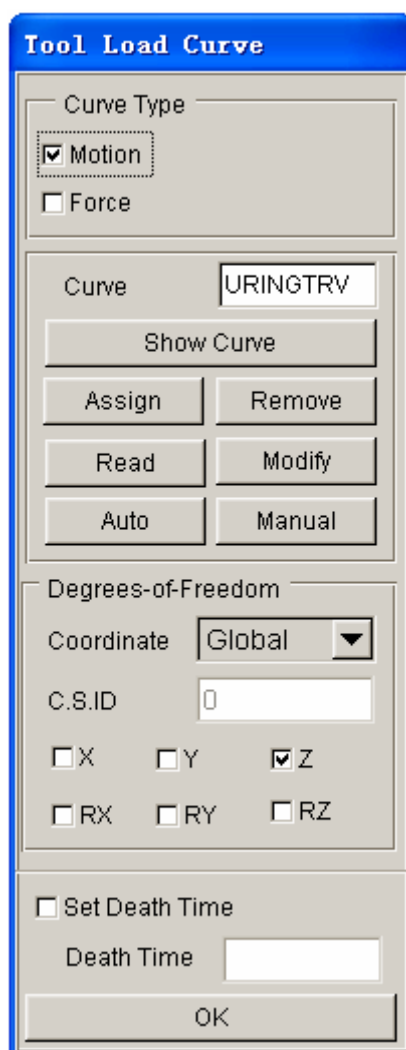


Figure 9.2.4 Tool Load Curve dialog box

#### SHOW CURVE

Display the load curve on the eta/DYNAFORM display area. It also provides many options for viewing the graph.



**ASSIGN CURVE**

This function is used to assign a motion/force load curve to a tool from the list of defined load curves in the SELECT CURVE dialog box. As shown in Figure 9.2.5, the user can select a particular load curve from the list. Click on the OK button to confirm the selection and to complete the ASSIGN CURVE operation.

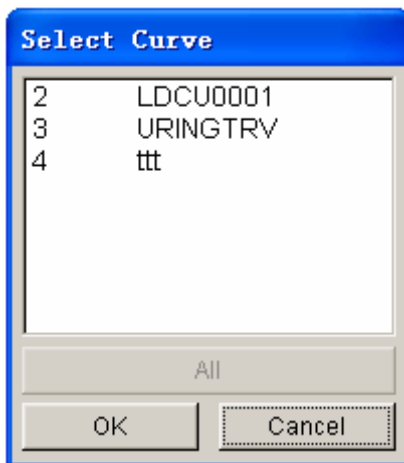


Figure 9.2.5 List Curve

**READ**

Allow the user to read in an external load curve using the window in Figure 9.2.6.

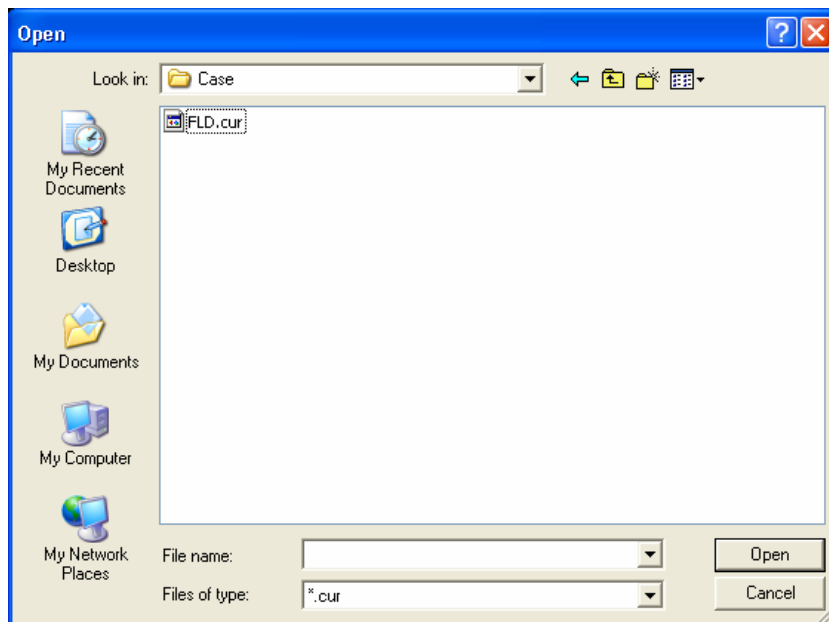


Figure 9.2.6 Read Curve File

**AUTO**

This function is used to generate a velocity/displacement/force load curve. If the MOTION option is checked, as illustrated in Figure 9.2.4, the MOTION CURVE dialog box as illustrated in Figure 9.2.7 is displayed.

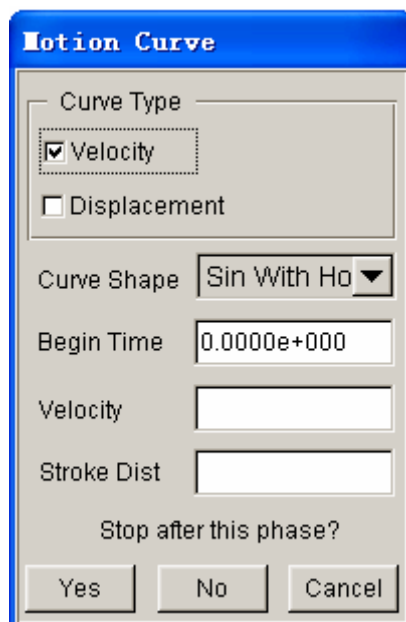


Figure 9.2.7 Auto Motion Curve dialog box

The MOTION CURVE dialog box enables the user to define either the velocity or the displacement load curve according to the begin time, velocity/displacement and stroke distance. If the VELOCITY checkbox is toggled on, the load curve is plotted as velocity versus time. The DISPLACEMENT option enables the user to plot a displacement versus time load curve. The load curves can be plotted in one of the 3 different types of curve shapes as listed below:

**TRAPEZOIDAL** – Simplified discrete load curve (recommended).

**SINUSOIDAL** – Smooth discrete load curve.

**SINUSOIDAL WITH HOLD** – Smooth discrete load curve with the maximum velocity held constant.

After selecting the desired curve shape, the user does the following steps:

- Input the data fields of BEGIN TIME, VELOCITY, and STROKE DISTANCE.
- Select NO if there is more than one travel phase in the curve, following by selecting curve shape and input the values for a second travel phase in the data fields accordingly.

- Select YES to create the motion curve. A dialog box showing the motion curve is displayed. Refer to Section 11.13.8, UTILITIES/LOADCURVE/SHOW LOAD CURVE.
- If FORCE CURVE is checked, the FORCE CURVE dialog box (as shown in Figure 9.2.8) is displayed.

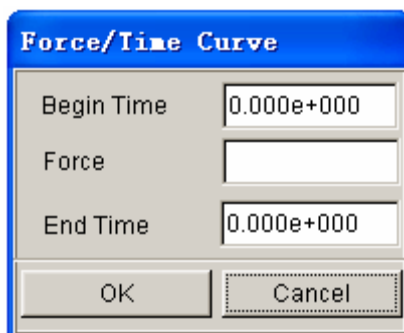


Figure 9.2.8 Auto Force Curve dialog box

- Input values in the data fields of BEGIN TIME, FORCE, and END TIME. The default end time is the end time of all motion curves.
- Select OK to create the force curve. A dialog box showing the force curve will show up. Refer to Section 11.13.8, UTILITIES/LOAD CURVES/SHOW LOAD CURVE.

### REMOVE

This function is used to remove the applied load curve. If the REMOVE button is clicked, a warning message (as shown in Figure 9.2.9) is displayed. The user selects either YES/NO to accept/reject the operation.

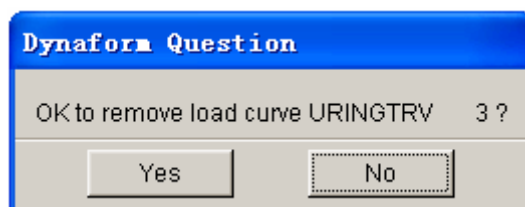


Figure 9.2.9 Remove Warning

### MODIFY

This function is used to edit the current curve. Refer to Section 11.13.4 for more details.

### MANUAL

This function is used to manually create a curve by input data points. The dialog box is shown in Figure 9.2.10.

The steps listed below is used to manually generate load curve:

- Select VELOCITY or DISPLACEMENT option.
- Input TIME and VALUE (velocity or displacement).
- Select ADD POINT to add next data points.
- Select OK to complete curve definition and exit the INPUT CURVE dialog box.

Time	Value
0.00	0.00
0.01	0.02

Figure 9.2.10 Input Curve dialog box

### 9.2.7 GENERATE A TOOL FROM MATING TOOLS

This function is used to generate the tool mesh from an existing tool. It combines the operations of copying or offsetting elements, creating parts, and adding parts to the current tool definition.

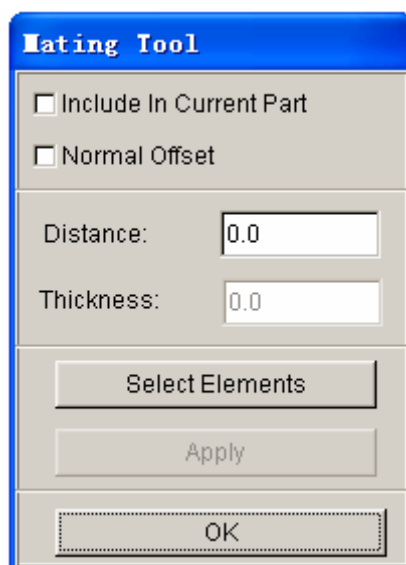


Figure 9.2.11 Mating Tool dialog box

- Select OFFSET FROM MATING TOOL to display the MATING TOOL dialog box (as shown in Figure 9.2.11).
- Toggle the INCLUDE IN CURRENT PART checkbox on/off. If the option is toggled on, the new copied or offset elements is included in the current part, which is included in the current tool definition. If the option is toggled off, a new part is automatically created and added to the current tool definition.
- The NORMAL OFFSET option is used to offset the elements in normal direction. If toggled on, the user can key in offset THICKNESS.
- The user can set the location of the new tool along the stroke direction by inputting DISTANCE. The distance is measured from the original tool mesh.
- To offset or copy elements, select APPLY.
- Select OK to exit the dialog box.

### 9.3 POSITION TOOLS

The POSITION TOOLS menu contains a submenu that offers the following operations:

- AUTO POSITION
- MOVE TOOL
- MIN. DISTANCE

### 9.3.1 AUTO POSITION

This function enables the user to automatically position the defined tools according to the user-defined draw type, tool clearance and the stroke direction. Refer to Section 9.1, ANALYSIS SETUP, to edit the default setting.

### 9.3.2 MOVE TOOL

This function allows the user to translate any tool in any current C.S (Global or User Defined) direction. The dialog boxes are shown in Figures 9.3.1 and 9.3.2 for Global and Local C.S, respectively. To move the tool, the user selects the target tool from the tool list, the direction, and keys in the distance for translation. Then, selects APPLY to complete the translation. The user can select REVERSE to translate the tool in the opposite direction.

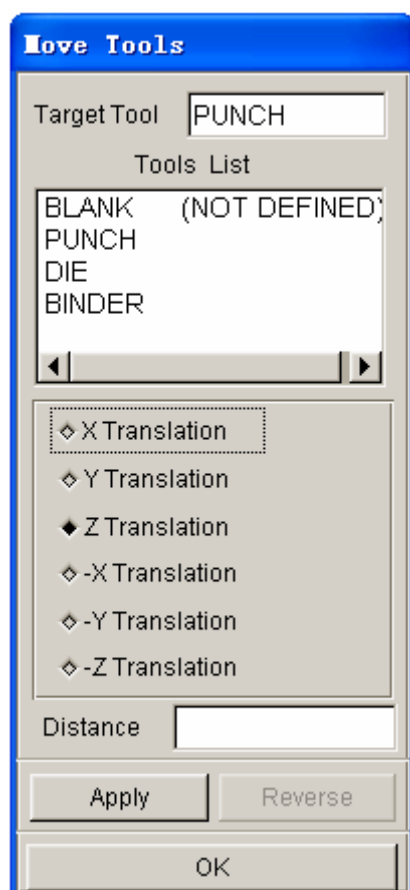


Figure 9.3.1 Global C.S. is Current C.S.

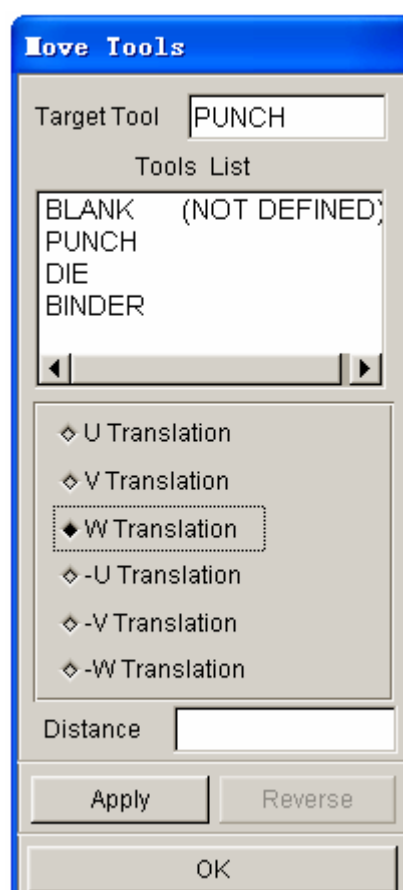


Figure 9.3.2 Local C.S. is Current C.S.

### 9.3.3 MIN. (MINIMUM) DISTANCE

This function allows the user to measure the minimum distance between two selected tools in the selected direction. To measure the minimum distance, the user has to select the MASTER and SLAVE TOOLS as shown in Figure 9.3.3.

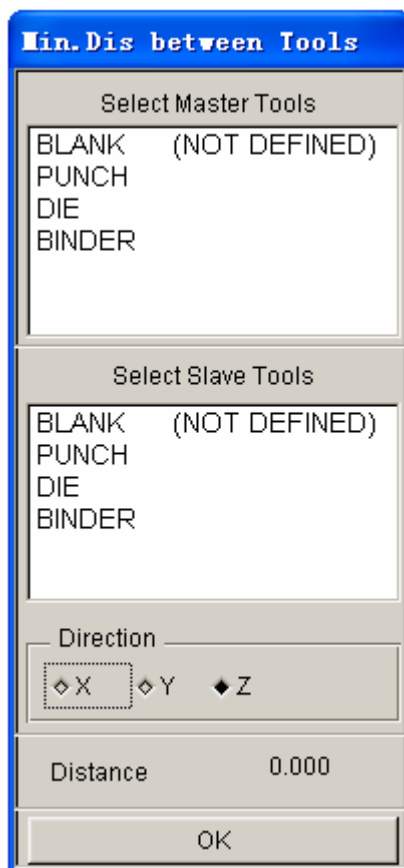


Figure 9.3.3 Min. Distance dialog box

## 9.4 DRAWBEAD (CTRL+D)

The functions in the DRAWBEAD menu enables the user to create, modify, and assign drawbeads. The dialog box is shown in Figure 9.4.1.

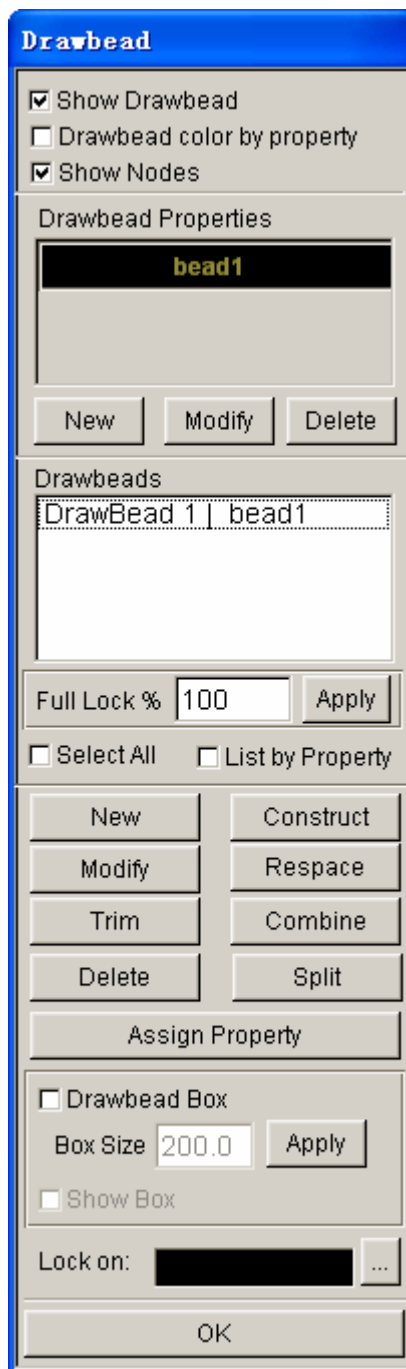


Figure 9.4.1 Define Drawbead dialog box

**9.4.1 DRAWBEAD DISPLAYING OPTION**

The user can select the following three options to display the drawbeads.

**9.4.1.1 SHOW DRAWBEAD**



If this option is toggled, the drawbeads are displayed on the screen.

### 9.4.1.2 DRAWBEAD COLOR BY PROPERTY

This feature allows the user to display the drawbead with either property color or the different color with different lines.

### 9.4.1.3 SHOW NODES

This feature allows the user to show the nodes of the drawbead. If this option is toggled on, all the nodes of the drawbeads are displayed on the screen.

## 9.4.2 CREATE PROPERTIES

In the eta/DYNAFORM, each drawbead carries its own property. Prior to creating the drawbead, the user must define the property accordingly. In the create property field, the NEW, EDIT, DELETE functions enable the user to create, modify and delete the property, respectively.

### 9.4.2.1 NEW PROPERTY

Click on the NEW button to display the DRAWBEAD PROPERTY dialog box as shown in Figure 9.4.2.

DRAW BEAD PROPERTIES	
SECTION TITLE	bead1
STATIC FRICTION COEF.	1.000000E-001
DYNAMIC FRICTION COEF.	0.000000E+000
BENDING LOAD CURVE ID	0
<input checked="" type="checkbox"/> Full Lock Force	0.000000E+000 <span>Default</span>
DRAW BEAD DEPTH	1.000000E+000
BENDING CURVE SCALE	1.000000E+000
<input type="button" value="OK"/> <input type="button" value="Advanced"/> <input type="button" value="Default"/> <input type="button" value="Reset"/> <input type="button" value="Cancel"/>	

Figure 9.4.2 Drawbead properties dialog box

### SECTION TITLE

Allow the user to name title for the property, the default title is bead1, bead2 and so on.

#### STATIC FRICTION COEF.

Allow the user to set the static coefficient of friction between the blank and the drawbead, the default static coefficient of friction is 0.1.

#### DYNAMIC FRICTION COEF.

Allow the user to set the dynamic coefficient of friction between the blank and the drawbead, the default dynamic coefficient of friction is 0. It means the dynamic friction between the blank and the drawbead is ignored.

#### BENDING LOAD CURVE ID

Allow the user to define the  $F_{bending}$  component of the drawbead restraining force. The  $F_{bending}$  component is referred as force per unit drawbead length as a function of displacement  $\delta$  as illustrated in Figure 9.4.3.

**NOTE:** *The NORMAL FORCE of drawbeads is set to 1.0e-6 automatically by eta/DYNAFORM. Hence, only the BENDING FORCE needs to be defined by the user.*

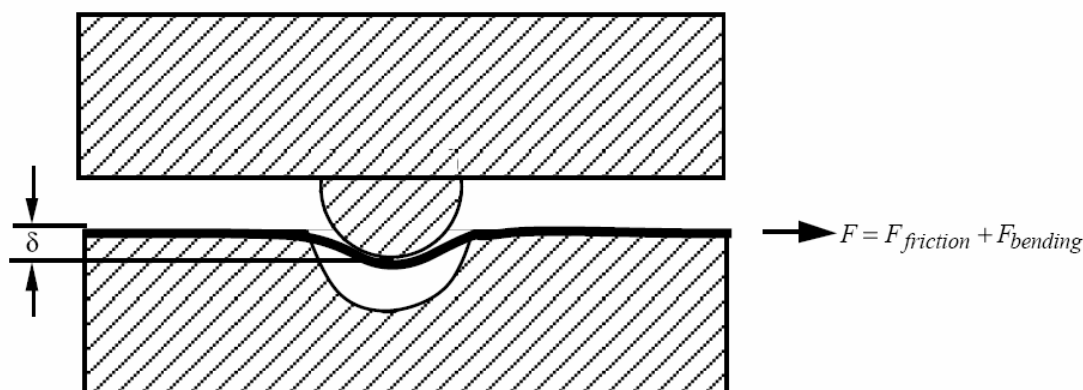



Figure 9.4.3 Drawbead resisting force as a function of the drawbead displacement

The user can click on the  button to define the bending curve for the property using the DRAWBEAD LOAD CURVE dialog as illustrated in Figure 9.4.4. The user can select one of the curves in the list or select the NEW LOAD CURVE functions to generate drawbead force curve using one of the following two methods:

**AUTO** – This function enables the user to create bending curve according to the drawbead section figure and the material of the blank as illustrated in Figure 9.4.5. The user can input all the parameters in the DRAWBEAD FORCE PREDICTION dialog box, following by clicking on the OK button to exit the dialog box. The defined curve ID is specified as the BENDING curve ID for the property.

**MANUAL** – This function enables the user to manually create a bending curve for the property using the function in the dialog box as illustrated in Figure 9.4.6. Please refer Section 12.12 for detailed information about load curve definition.

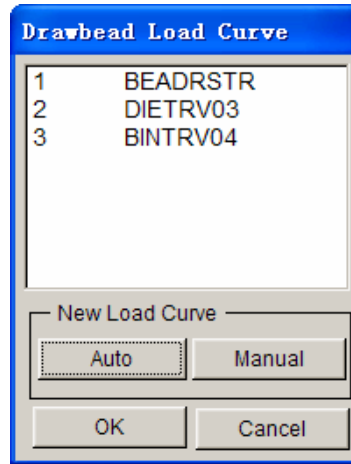


Figure 9.4.4 Drawbead Load Curve dialog box

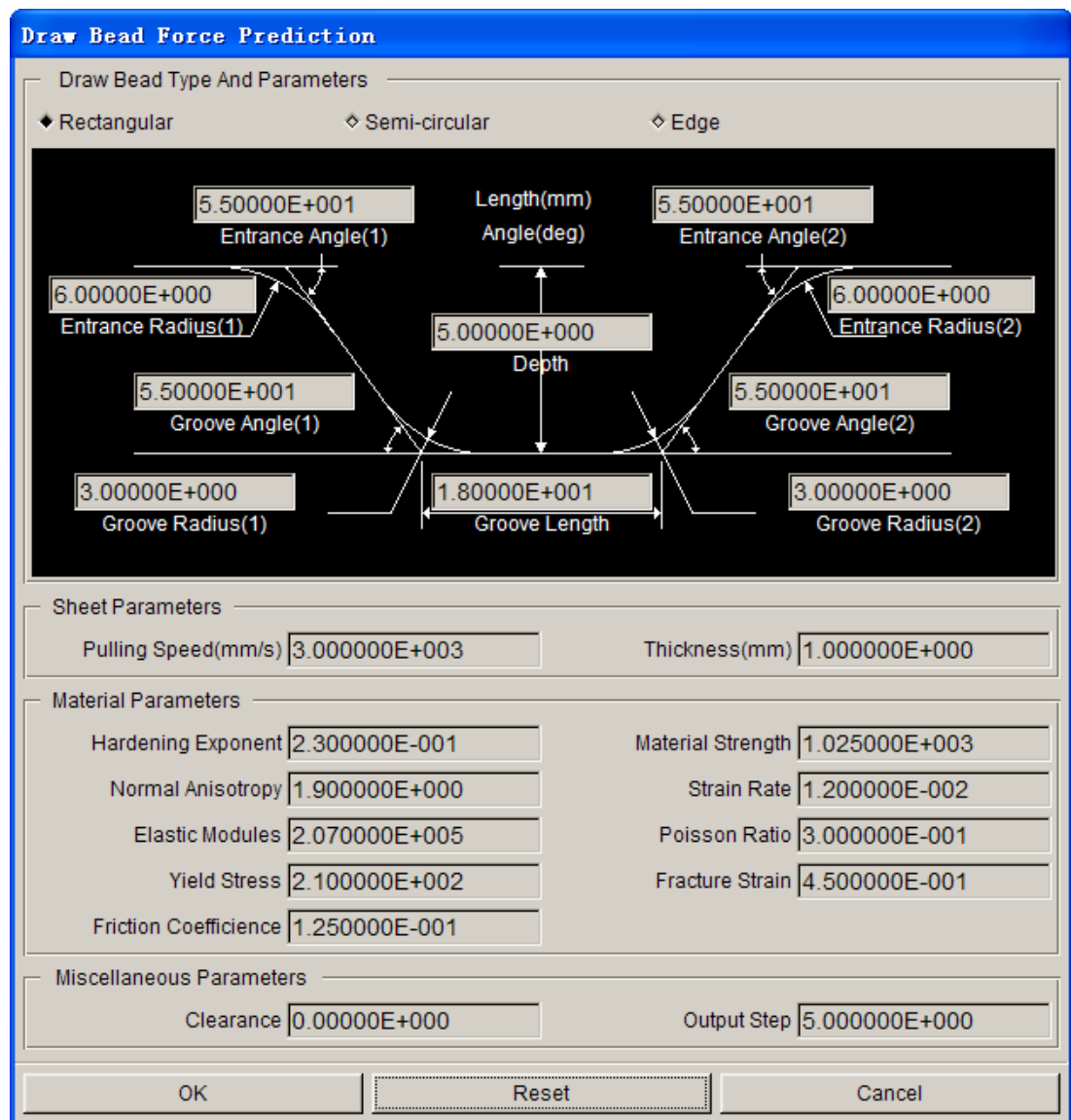


Figure 9.4.5 Drawbead Force Prediction dialog box

X	Y
0	0
0.01	1

Figure 9.4.6 Define the Bending curve

**FULL LOCK FORCE**

This is an optional method to define the BENDING FORCE for the drawbead. If the checkbox is toggled on, the user can manually key in the FULL LOCK FORCE value in the input data field or click on the DEFAULT button to calculate the default full lock force. Then, the BENDING FORCE will calculate automatically according to the FULL LOCK FORCE.

*NOTE: The default FULL LOCK FORCE is calculated by the material ultimate tensile strength and the thickness of the blank. The user has to define the blank material and properties prior to obtaining the default FULL LOCK FORCE.*

**DRAWBEAD DEPTH**

Allow the user to define the depth of the drawbead, The default depth of drawbead is set as 1.0 (mm).

**BENDING CURVE SCALE**

Allow the user to set the scale factor for the BENDING CURVE. The default scale factor is set as 1.0 (or 100%).

After the user creates each new property, the new property is added to the property list and set as the current property.

**9.4.2.2 MODIFY PROPERTY**

This function enables the use to edit the current property.

### 9.4.2.3 DELETE PROPERTY

This function enables the use to remove the current property.

*NOTE: The current property can be removed only if the current property is not assigned to any drawbead. Else, the following message is printed in the message prompt window:*

***THE PROPERTY ASSIGNED TO DRAWBEAD. CAN NOT BE DELETED.***

### 9.4.3 CREATE DRAWBEAD

After a drawbead property is defined, the user can continue with generation of drawbead by clicking on the NEW DRAWBEAD button as illustrated in Figure 9.4.7. In the DRAWBEADS field, there are some functions such as RESPACE, TRIM, COMBINE, DELETE, SPLIT, which are utilized to edit the drawbead. The description of each function is detailed in the following sections.

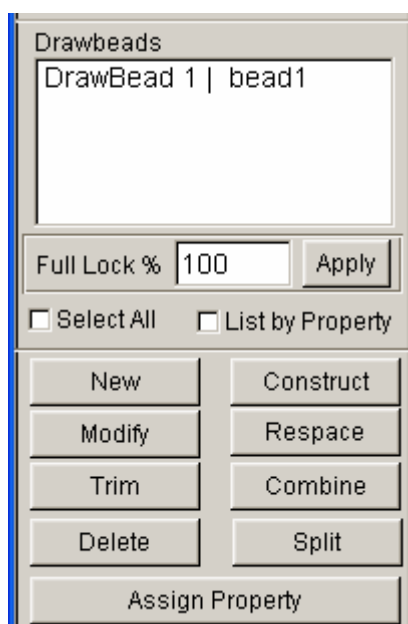


Figure 9.4.7 Drawbeads field

#### 9.4.3.1 NEW DRAWBEAD

This function enables the user to create drawbead through selecting the existing geometrical lines. After clicking the NEW DRAWBEAD button, the SELECT LINE dialog box as illustrated in Figure 9.4.8 is shown. Then the user can select one or more lines to define the drawbead.

At the same time, the function enables the user create drawbead through selecting the nodes. Click the CONSTRUCT button to pop up the SELECT NODE dialog box. The user can refer to section 5.4.3 about how to select nodes. After selected the nodes, the program will automatically construct a curve by selected nodes, and then create drawbead through the curve.

**NOTE:** *Prior to creating a new drawbead, the user has to select one property from the drawbead property field. The selected property is assigned as the property for new drawbead.*

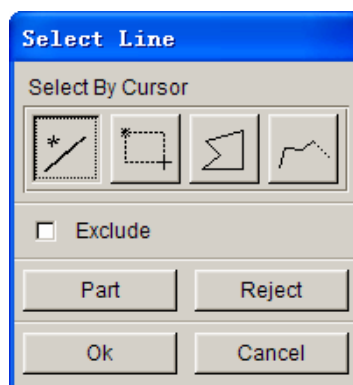


Figure 9.4.8 Select Line dialog box

The DRAWBEAD INPUT dialog box as illustrated in Figure 9.4.9 is displayed after a line is selected. The function provided in the dialog box enables the user to set the density of point on each drawbead line.



Figure 9.4.9 Define the density of points on the drawbead line

#### **BY DISTANCE**

Define the points on line by setting the constant distance between two adjacent points on line. The default distance is 20 (mm).

**UNIFORM**

The specified number of points is evenly spread on the line. The default number of points is set as 20 on each line. Refer to Figure 9.4.10a for drawbead line with uniform point distribution.

**CHORDAL DEVIATION**

The points on the lines are distributed according to the CHORDAL DEVIATION value. For drawbead line with larger curvature, more points are distributed. If the CHORDAL DEVIATION value is small, more points are created on the drawbead line. The default CHORDAL DEVIATION is set as 0.25. Refer to Figure 9.4.10b for drawbead line with point distribution based on CHORDAL DEVIATION.

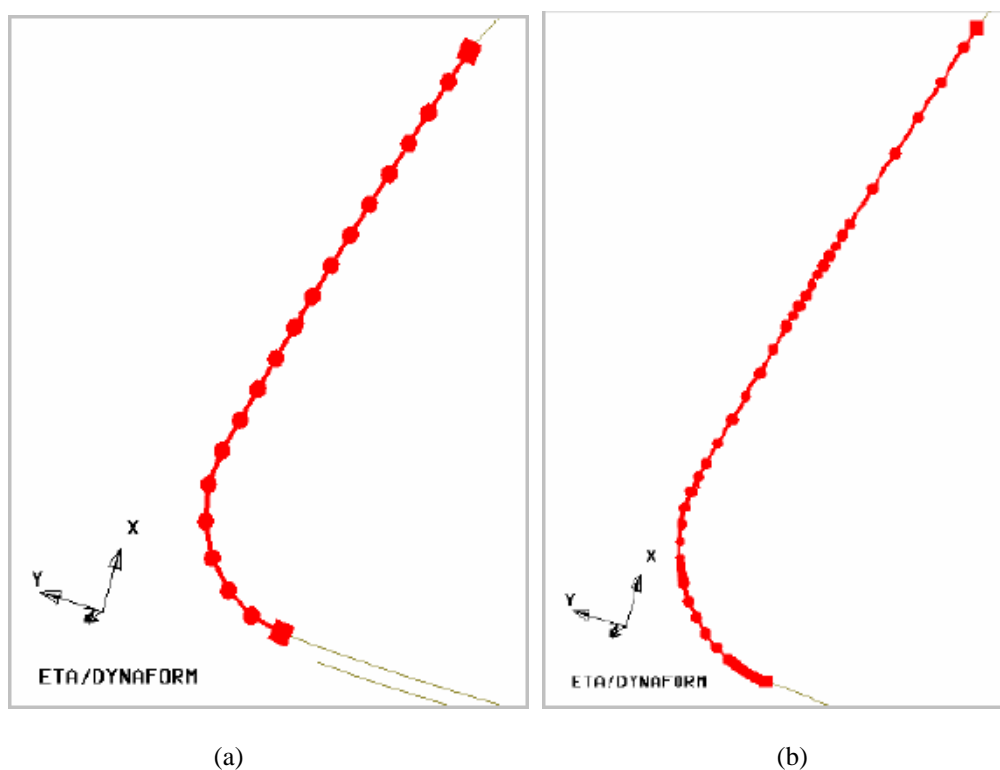
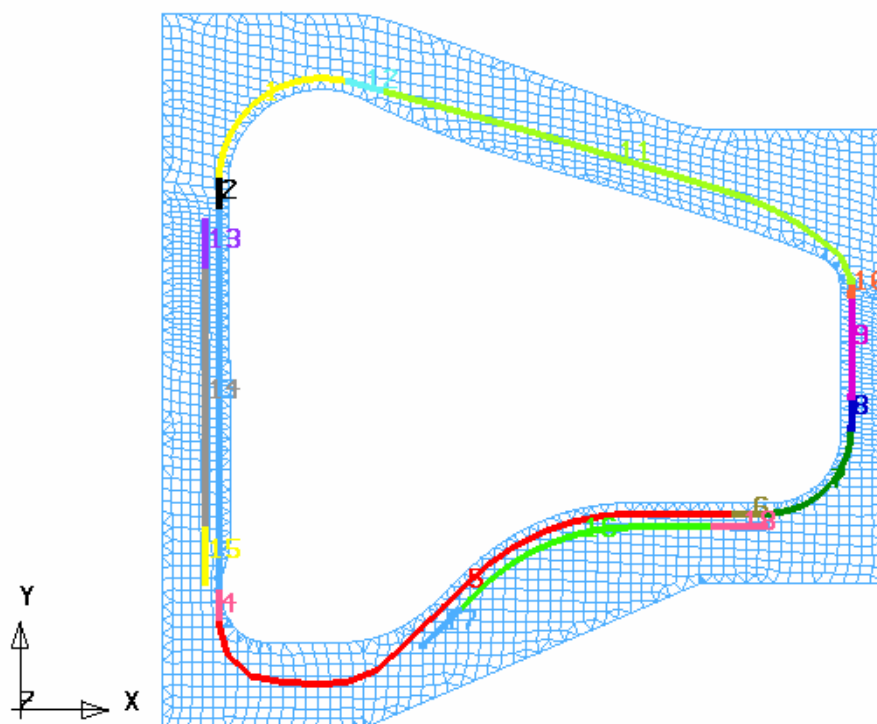


Figure 9.4.10 The distribution of points on drawbead line (a) Uniform (b) Chordal deviation

The user can select more than one drawbead line as drawbead using the same property. Alternatively, the user can define different line using different property.

The drawbead is created on each selected line. Each drawbead in the Drawbeads field (as shown in Figure 9.4.7) is identified with a unique name such as Drawbead 1, Drawbead 2 and so on. As illustrated in Figure 9.4.11, the corresponding ID number is displayed on the screen beside the drawbead.





### ETA/DYNAFORM

Figure 9.4.11 Drawbead name

### DRAWBEAD LIST

All defined drawbeads are listed in the Drawbead List as illustrated in Figure 9.4.12. Each time a new drawbead is created, it is automatically included in the Drawbead List. The Drawbead List consists two columns. The left column displays the drawbead name while the right column is the property name corresponded to the drawbead. The user can select multiple drawbeads from the list by pressing the SHIFT or the CTRL key while simultaneously clicking the left mouse button to pick drawbeads. The selected drawbeads is highlighted both on the dialog box (in blue) and on the screen (in white).

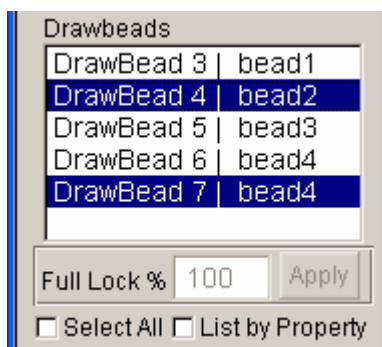


Figure 9.4.12 Drawbead List

**PERCENT**

This option enables the user to modify the lock force percentage for the selected drawbead(s). First, the user picks the drawbeads from the drawbead list. The key in the desired percentage following by clicking on the APPLY button to complete the modification.

**SELECT ALL**

Allow the user to select all the drawbeads on the DRAWBEAD LIST.

*NOTE: If the LIST by PROPERTY option is checked, all the drawbeads sharing the current property are selected.*

**LIST BY PROPERTY**

If this option is selected, only the drawbeads sharing the current property are displayed on the DRAWBEAD LIST. Please see the Figure 9.4.13.

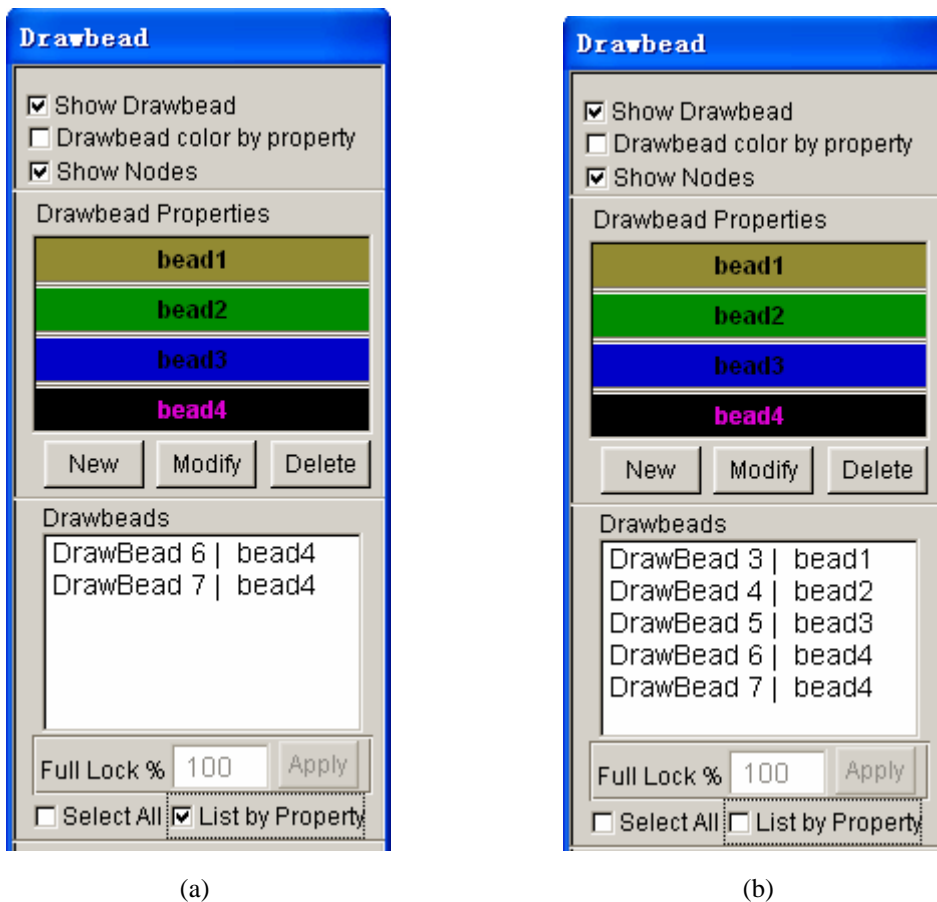


Figure 9.4.13 List by Property option (a) Check (b) Uncheck

**9.4.3.2 MODIFY**

This function enables the user to modify drawbead. Firstly, the user can select a drawbead, and then edit the curve in the pop-up CURVE EDIT dialogue box. Refer to correlative section about CURVE EDIT function.

### 9.4.3.3 RESPACE

This function enables the user to modify the density of points on the drawbead line using the DRAWBEAD INPUT dialog box as illustrated in Figure 9.4.9. Prior to performing the operation, the user has to select a drawbead from the drawbead list.

### 9.4.3.4 TRIM

This function enables the user to divide a drawbead along the original referred line. The user has to select a drawbead prior to performing such operation.

1. Select ENDPOINT of the drawbead that need to be TRIMMED. See the Figure 9.4.14.
2. Select a point on the Drawbead line as destination point

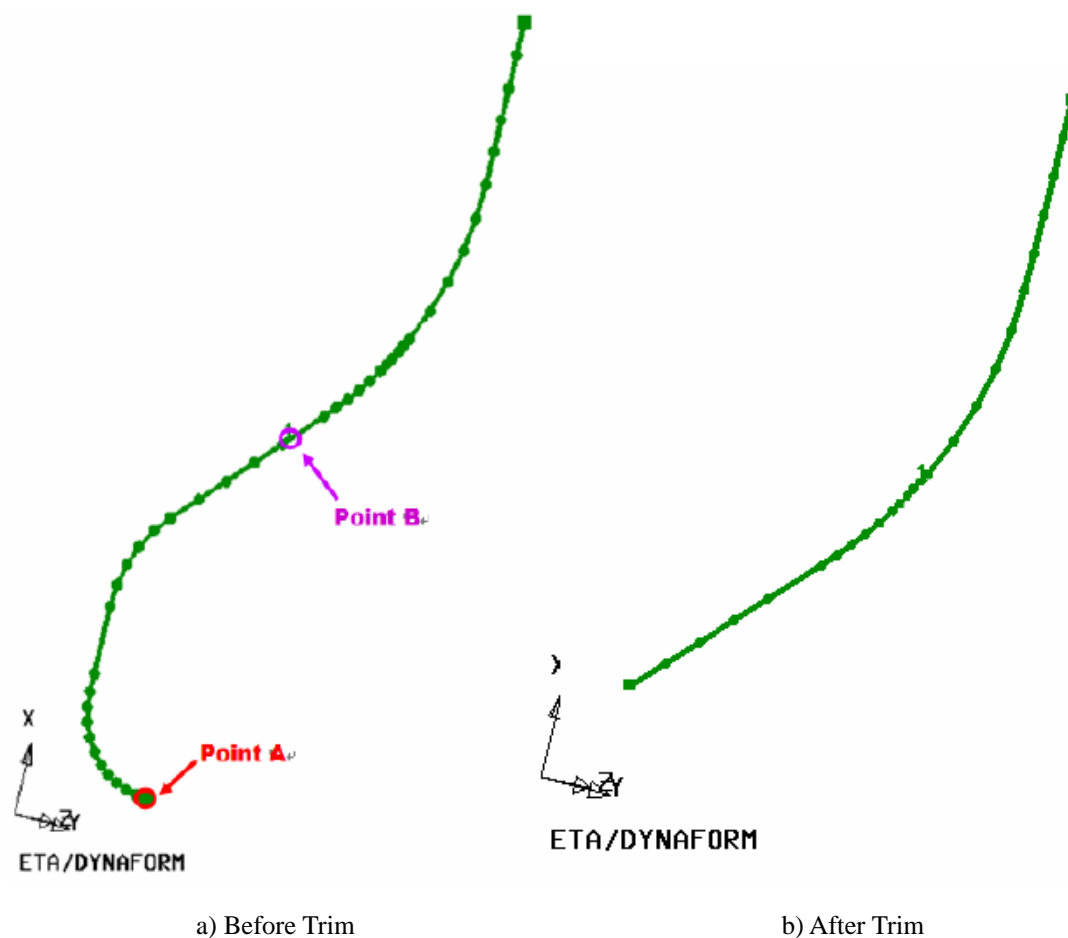


Figure 9.4.14 Trim the Drawbead

**NOTE:** The *TRIM* function also can be utilized to recover the *TRIMMED* drawbead along the original drawbead line. For example, a drawbead as illustrated in Figure 9.4.15 has been *TRIMMED*. The user can recover the trimmed drawbead.

1. Select the **END POINT A** as illustrated in Figure 9.4.15.
2. Select a point **B** on the original drawbead line as the destination point.

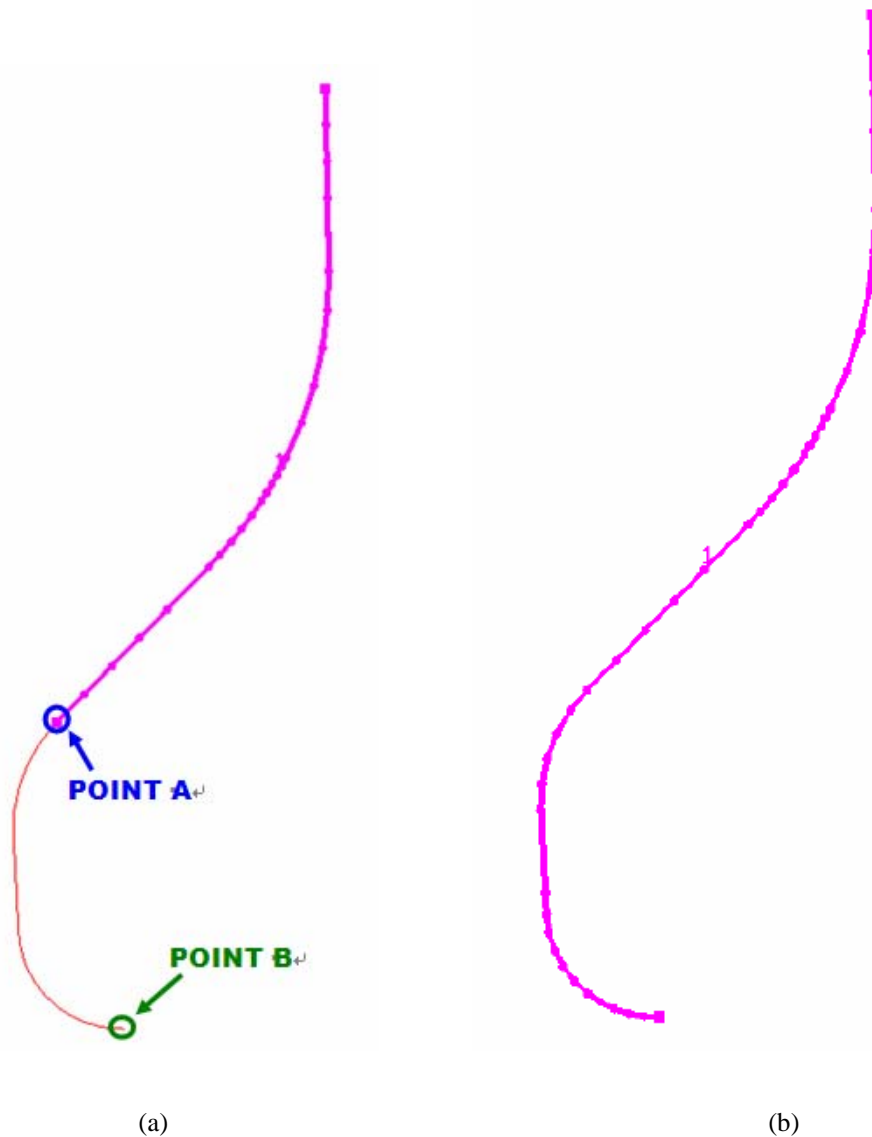


Figure 9.4.15 Trim Drawbead (a) Before trim (b) After trim

#### 9.4.3.5 COMBINE

This function enables the user to combine two adjacent drawbeads into one drawbead. To combine two

adjacent drawbeads, the user has to click on the COMBINE button, following by selecting two adjacent drawbeads using the mouse cursor. Figure 9.4.16 illustrates the COMBINE drawbead operation.

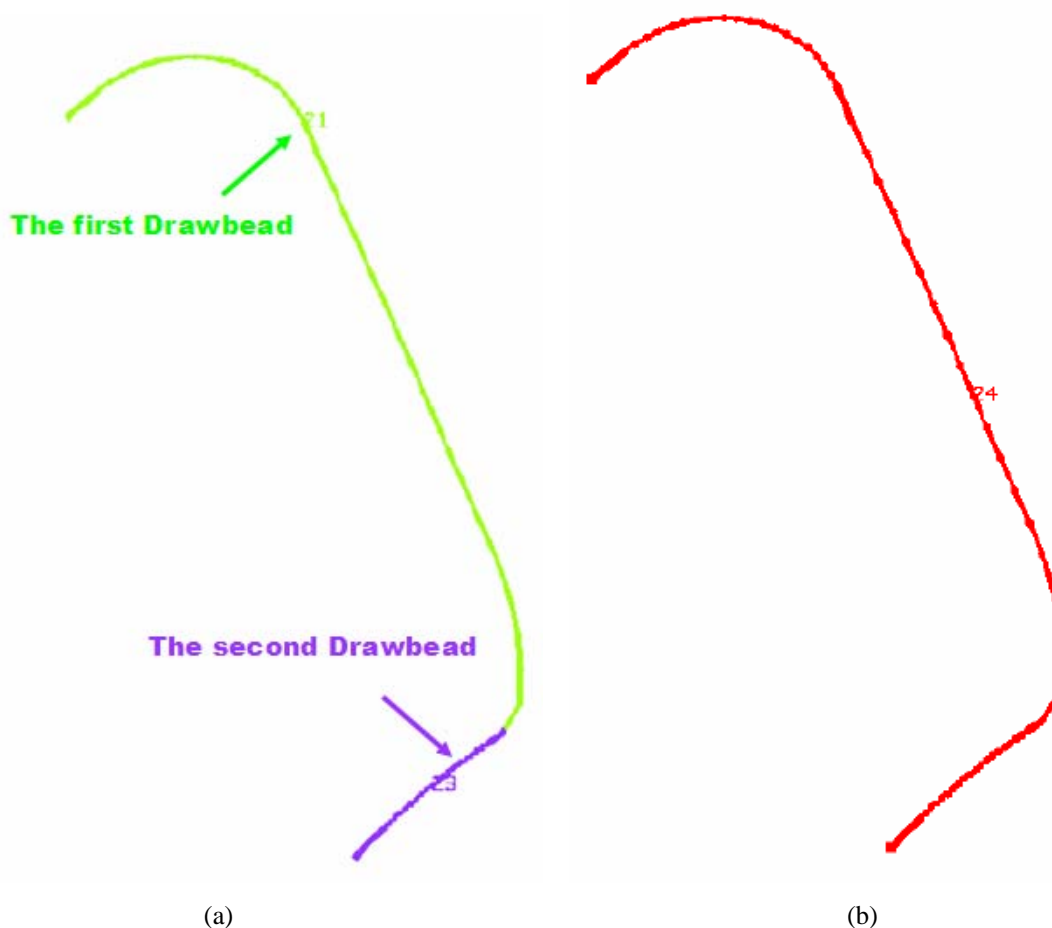


Figure 9.4.16 Combine drawbead (a) Before (b) After

**NOTE:** This function enables the user combine two adjacent drawbeads without the same property. If the two adjacent drawbeads have the same property, then the property of combined drawbead is the same as the selected drawbeads. If the two adjacent drawbeads have different property, then the property of combined drawbead is the same as the **FIRST** selected drawbead.

#### 9.4.3.6 DELETE

This function enables the user to remove the selected drawbeads. If the user has selected drawbead on the Drawbead list, then all the selected drawbeads are removed after he/she clicks on the DELETE button. If the user hasn't selected the drawbead, then a message is printed in the message prompt window asking the user to pick drawbead. The user can place the mouse cursor near the drawbead following by clicking on left mouse button to complete the DELETE drawbead operation.

### 9.4.3.7 SPLIT

This function enables the user to split a drawbead at selected location. After the function is activated, the user selects the desired location to split the drawbead using the mouse cursor. eta/DYNAFORM identifies the nearest NODE to the selected location and splits the drawbead into two drawbeads. The drawbeads share the same property with the original drawbead. Figure 9.4.17 illustrates SPLIT drawbead operation.

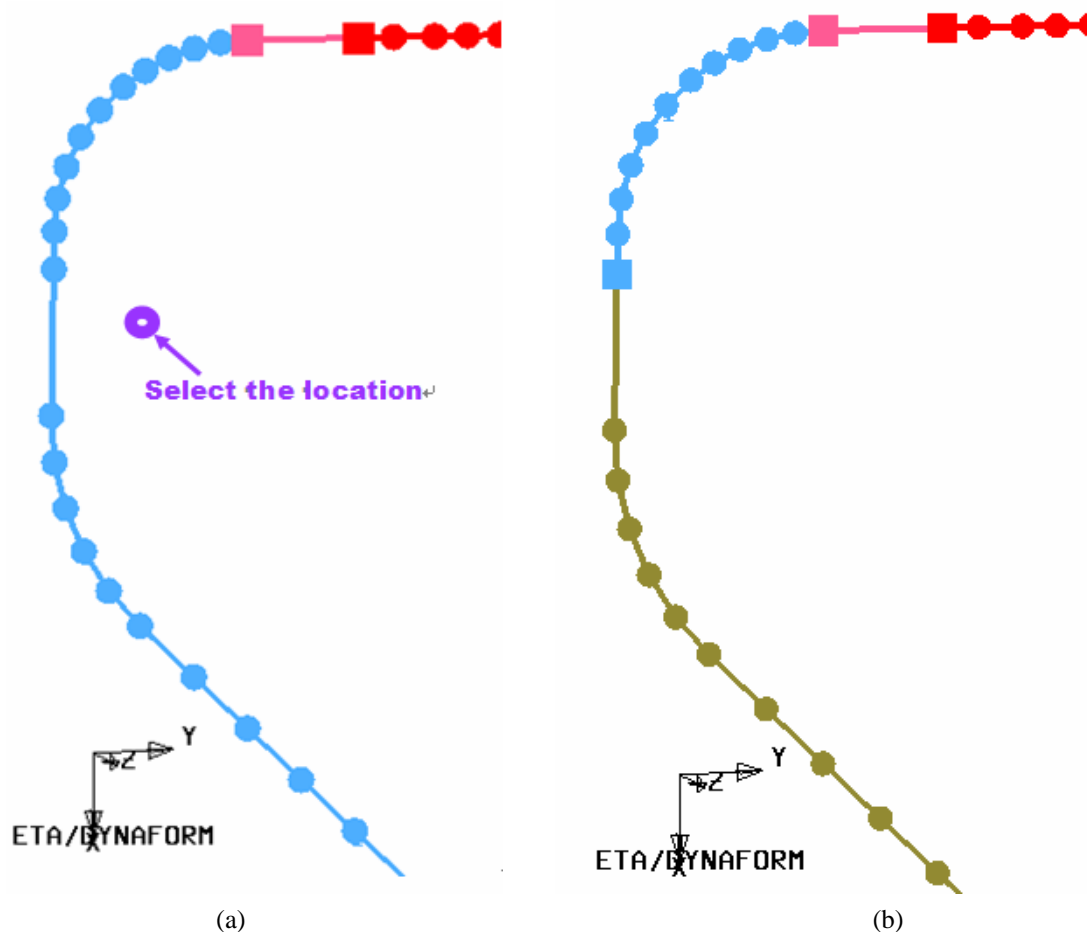


Figure 9.4.17 Split drawbead (a) Before (b) After

### 9.4.4 DRAWBEAD BOX

The DEFINE BOX function is used to define a specific box-shaped volume around the drawbead. The box contains the drawbead nodes and elements between the bead and the outer edge of the blank. It is implemented to limit the size of the master surface for the drawbead contact. The BOX SIZE is the linear distance between the drawbead line and the surfaces of box. The default size of the drawbead box is 200. The user can key in the size of the box following by clicking the APPLY to modify the size of drawbead box.

**NOTE:** It is highly recommended to define a box around the drawbead to limit the size of the master surface (Blank) considered for the drawbead. It will substantially reduce cost and memory requirements for stamping simulation.

#### 9.4.5 SHOW BOX

The option enables the user to show the drawbead box on the display area. The user can visualize and check the size of the box around each drawbead. Figure 9.4.18 illustrates drawbead boxes.

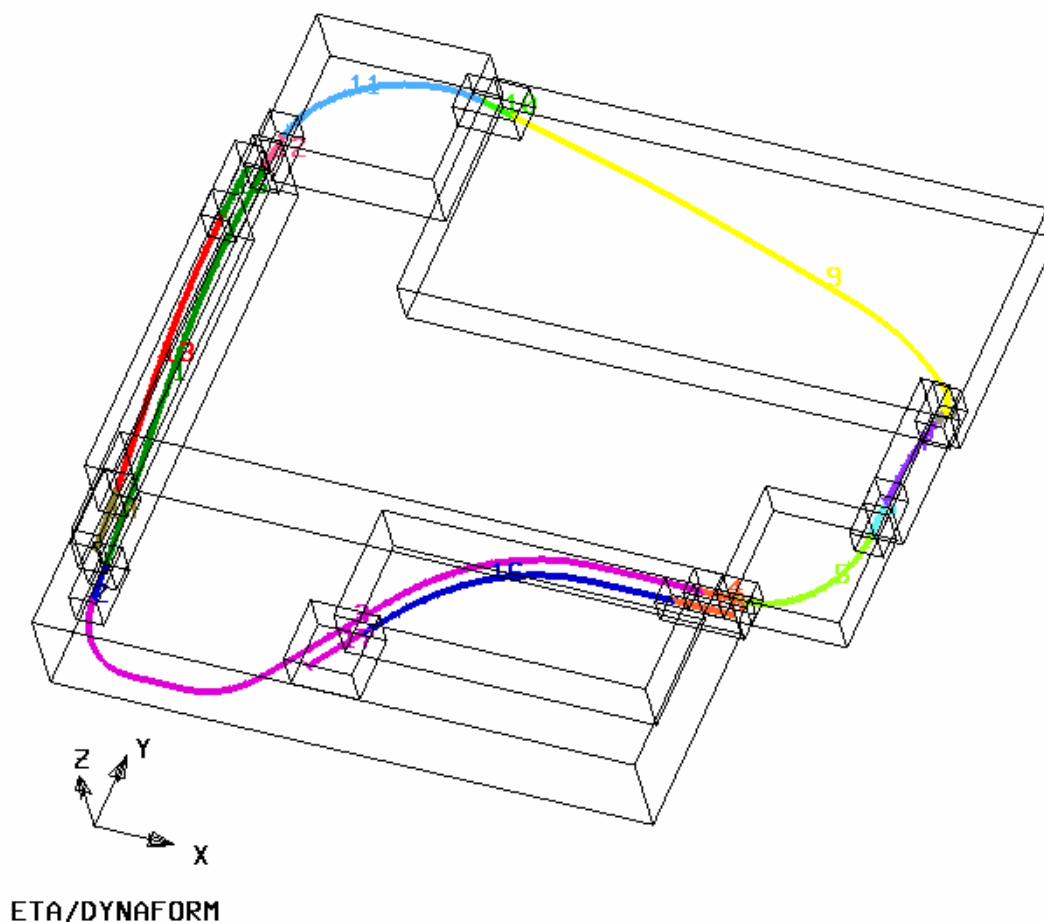



Figure 9.4.18 Show Box

#### 9.4.6 LOCK ON DRAWBEAD

The function enables the user to project the selected drawbead(s) on a part along the stamping direction or the working direction of the part. Prior to implementing the function, the user should pre-select the

drawbead(s). Then, click on the  button to display the SELECT PART dialog box as illustrated in Figure 9.4.19. The user selects a target part upon which the drawbead(s) will be locked. Normally, the DIE or BINDER is selected as target part.

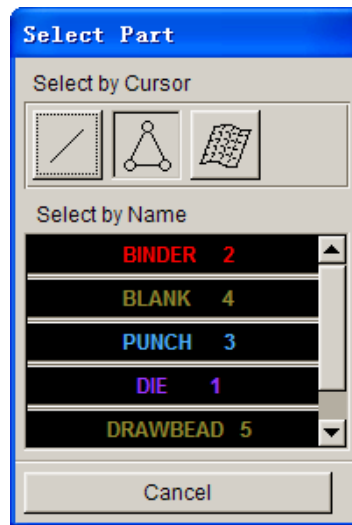


Figure 9.4.19 Select Part dialog box

After selecting the target part, a eta/DYNAFORM QUESTION dialog box as illustrated in Figure 9.4.20 is displayed. The user can select YES to project the drawbead(s) on the target part. Else, the user can select NO to cancel the operation.

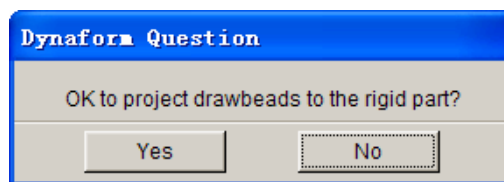


Figure 9.4.20 DYNAFORM Question dialog box



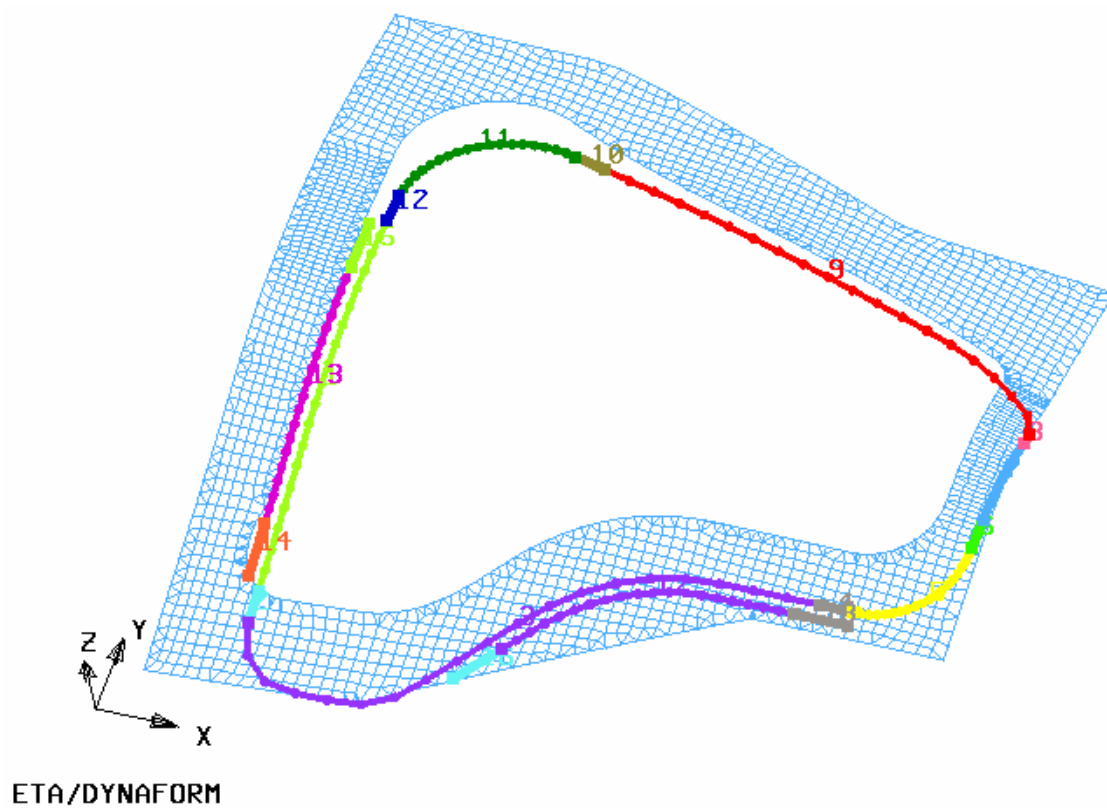


Figure 9.4.21 Before locking drawbeads on BINDER

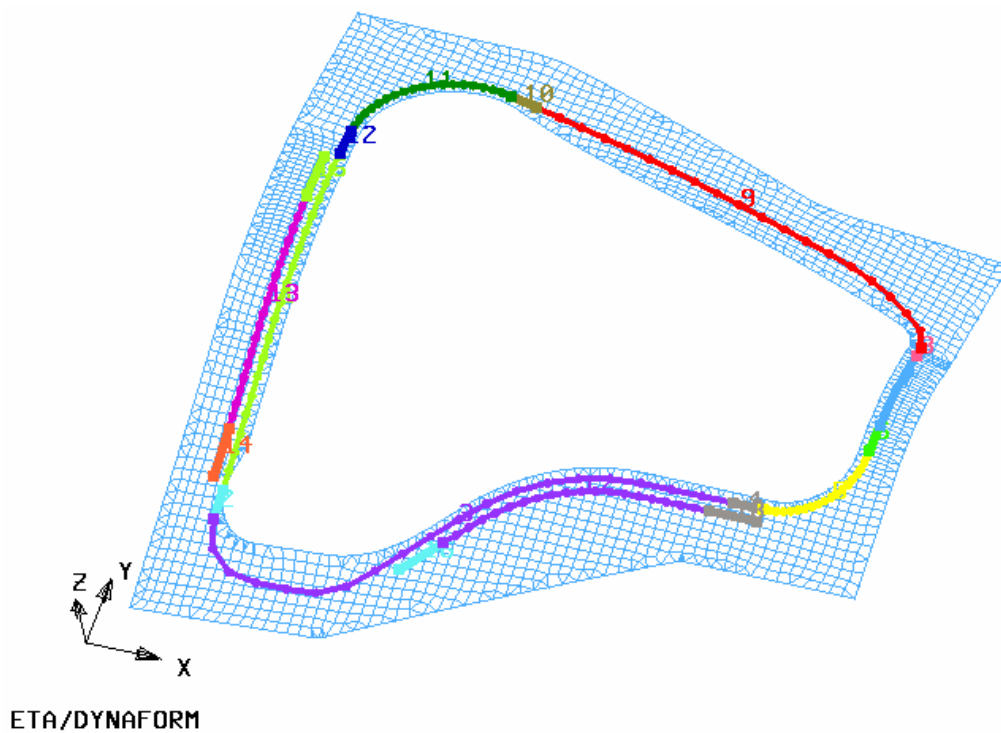


Figure 9.4.22 After locking drawbeads on BINDER

After the drawbead is locked on a target part, the name of target part is displayed in the LOCK ON box (as illustrated in Figure 9.4.23) whenever the user selects a drawbead from the Drawbead list.

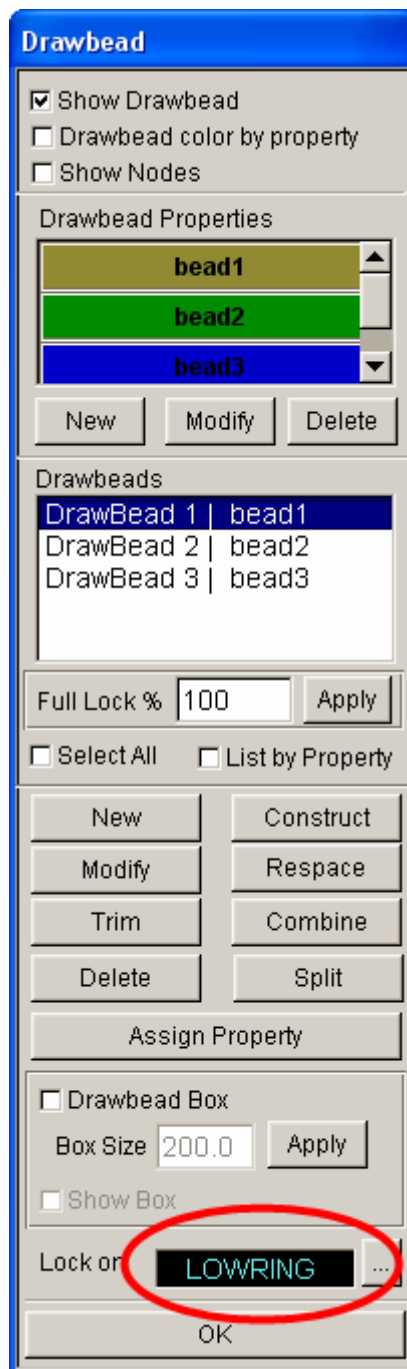


Figure 9.4.23 Lock on Part

**NOTE:** If the user defines the drawbead through the menu Setup/AutoSetup/Tools/Drawbeads, the drawbeads are projected onto the target part along its WORKING DIRECTION.

After all the parameters are defined, the user can click on the OK button to exit the DRAWBEAD dialog box. If some drawbeads are not locked on rigid part, a dialog box as illustrated in Figure 9.4.24

is displayed.

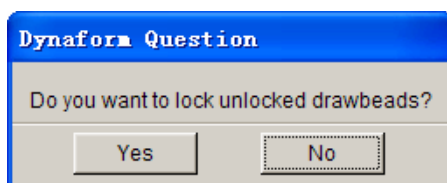


Figure 9.4.24 eta/DYNAFORM Question dialog box

The user can select YES to project the drawbead(s) on the target part. Else, select NO to exit the DRAWBEAD dialog box without locking these drawbead(s) on the target part.

## 9.5 BLANK GENERATOR

This function is mainly utilized for meshing flat blanks and tube. The dialog box is shown in Figure 9.5.1.

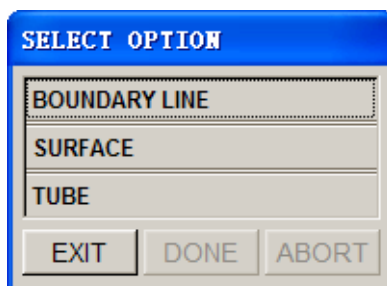


Figure 9.5.1 Blank Generator Options

### 9.5.1 BOUNDARY LINE

This function enables the user to create a flat blank mesh using boundary lines. After selecting the function, the user has to select the boundary lines using the functions provided in the SELECT LINE dialog box. Upon completion of line selection, the MESH SIZE dialog box illustrated in the Figure 9.5.2 is displayed.

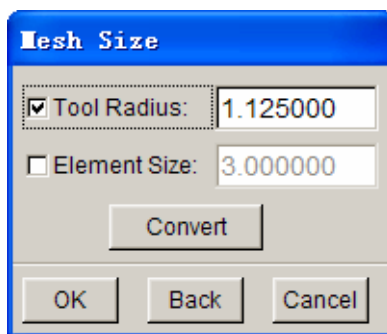


Figure9.5.2 Mesh Size dialog box

**TOOL RADIUS** – This input box enables the user to input the smallest concerned tool radius during the forming operation. Normally, it is the profile radius of DIE. The default Tool Radius is set as 1.125 (mm).

**ELEMENT SIZE** – This input box enables the user to input the average elements size for the blank mesh. The default Element Size (edge length) is set as 3.0 (mm).

**CONVERT**– This function enables the user to convert the Tool Radius to the Element Size or vice versa.

**OK**– This function enables the user to accept the defined parameters and continue the mesh operation. An eta/DYNAFORM QUESTION dialog box as illustrated in Figure 9.5.3 is displayed after the user select OK.

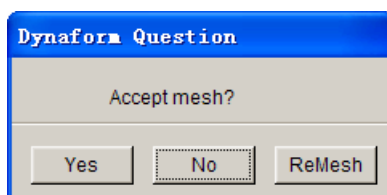


Figure 9.5.3 eta/DYNAFORM Question dialog box

**YES** – Accepts the blank mesh.

**NO** – Cancels the operation.

**REMESH** – The MESH SIZE dialog box is displayed again. The user can modify the radius/element size.

**NOTE:**

*1. The selected boundary lines should be lying on the same plane. If not, blank mesh is generated on the least square plane of the selected line.*

*2. The orientation of the layout of the selected line in the X-Y plane is important for the mesh. The user is recommended to rotate the lines along the Z-axis prior to generating the blank mesh. Please note that different blank mesh is generated according to the different line orientation.*

### 9.5.2 SURFACE

This function enables the user to create a flat blank mesh by selecting an existing surface. The operation is quite similar to blank mesh generation using the boundary lines.

### 9.5.3 TUBE

This function enables the user to create the tubular mesh for tubular hydroforming and tube bending simulation. After TUBE function is selected, the TUBE PARAMETER dialog box as illustrated in Figure 9.5.4 is displayed.

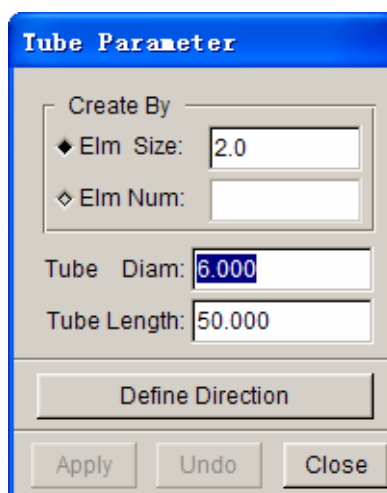


Figure 9.5.4 Tube Parameter dialog box

**ELEMENT SIZE** – Enables the user to input the average element size of tube mesh. The default element size is set as 2.0.

**ELEMENT NUMBER** – Enables the user to input the number of elements along the perimeter of tube.

**TUBE DIAMETER** – Enables the user to input the diameter of tube.

**TUBE LENGTH** – Enables the user to input the overall length of tube.

**DEFINE DIRECTION** – Enables the user to define the direction of tube mesh.

## 9.6 GUIDE & PIN

This function allows the user to create GUIDE and/or PIN in the eta/DYNAFORM. The guide and locator pin are utilized in metal stamping to prevent the blank from sliding during the binder wrap. The GUIDE & PIN dialog box is illustrated in Figure 9.6.1.

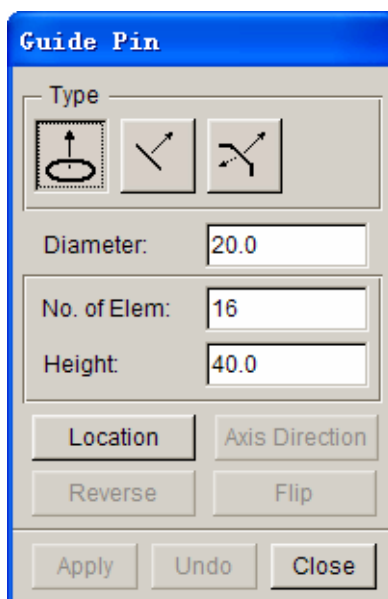


Figure 9.6.1 Guide and Pin dialog box

**DIAMETER** – Enables the user to define the diameter of locator pin. The default diameter is set as 20 (mm).

**NO. OF ELEM.** – Enables the user to define the number of element along the circumference of locator pin or the number of element along the length of guide.

**HEIGHT** – Enables the user to define the height of the locator pin or guide.

There are three types of guides offered in eta/DYNAFORM: Locator, Guide One and Guide Two.

### 9.6.1 LOCATOR

It is a straight column form pin. An illustration of locator pin is shown in Figure 9.6.2.

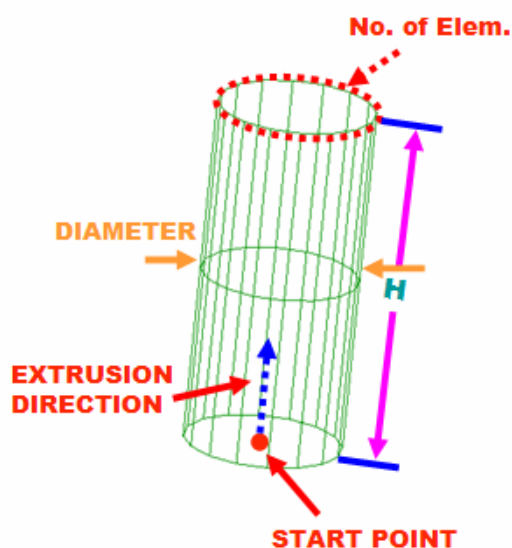


Figure 9.6.2 Locator Pin

**LOCATION** – This function enables the user to specify the location of the locator pin. After clicking on LOCATION button, the INPUT COORDINATE dialog box is displayed. The user has to select a point or node as location.

**AXIS DIRECTION** – This function enables the user to specify the direction of extrusion for the locator pin. After the location is defined, a highlighted arrow is displayed on the specified location denoting the direction of extrusion. The default orientation is the global +Z direction or the previously defined direction. The DIRECTION button is activated. The user can edit the direction of extrusion by hitting this button. The VECTOR dialog box as illustrated in Figure 9.6.3 is displayed to enable the change of direction.

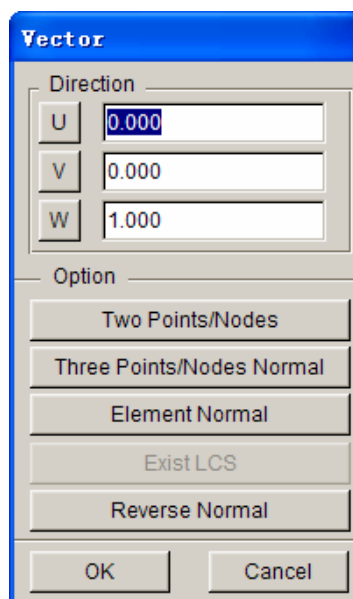


Figure 9.6.3 Vector dialog box

Please refer to Section 8.2 for more information about definition of vector direction.

**REVERSE** – This function enables the user to reverse the locator pin. An example is illustrated in Figure 9.6.4.

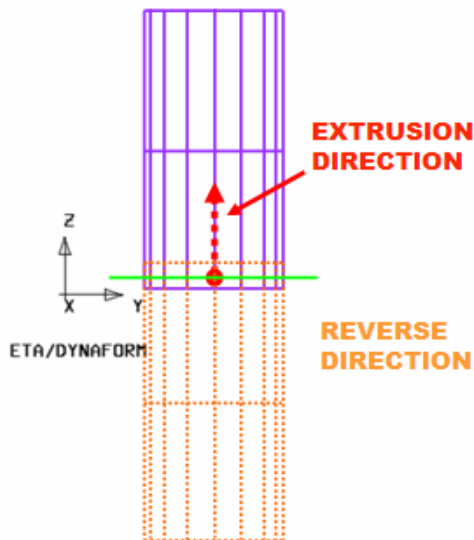


Figure 9.6.4 Reverse Locator Pin

**9.6.2 GUIDE ONE**

It is the straight guide without flange. An illustration of Guide One is shown in Figure 9.6.5.

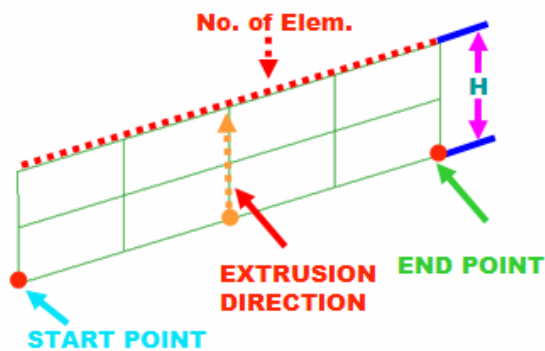


Figure 9.6.5 Guide One

The user should define the location and direction of extrusion to generate the guide. The location is specified by start and end point/node, as illustrated in Figure 9.6.5.

**9.6.3 GUIDE TWO**



It is the straight guide with flange. An illustration of Guide Two is shown in Figure 9.6.6.

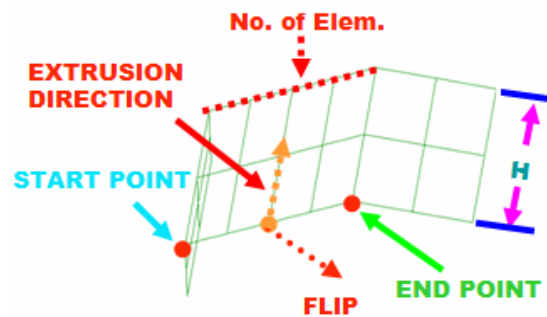


Figure 9.6.6 Guide One

Similarly to Guide One, the location and the direction of extrusion should be defined. Moreover, the user can modify the direction of the flange through the FLIP and REVERSE buttons. An illustration of Guide Two with REVERSE and FLIP result is shown in Figure 9.6.7 and Figure 9.6.8, respectively.

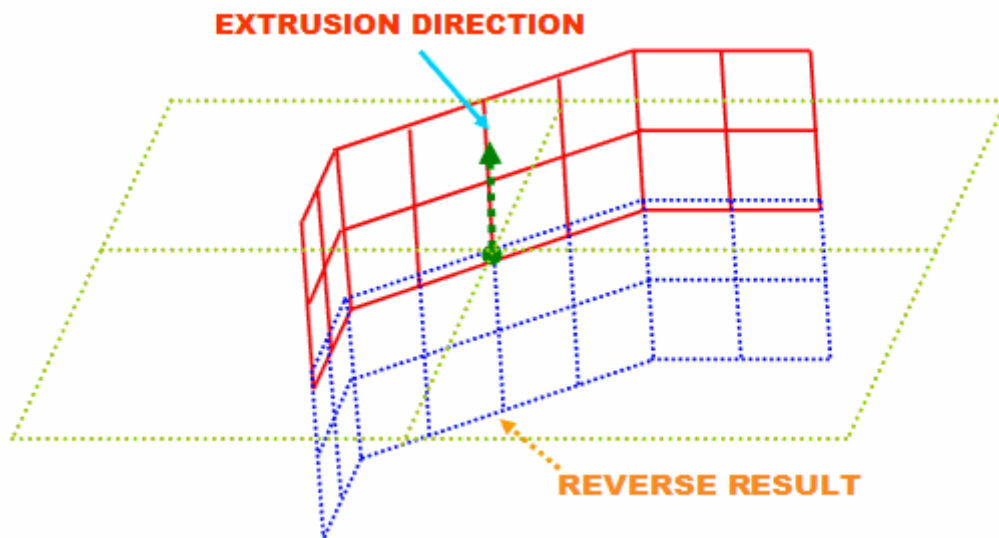


Figure 9.6.7 Reverse result

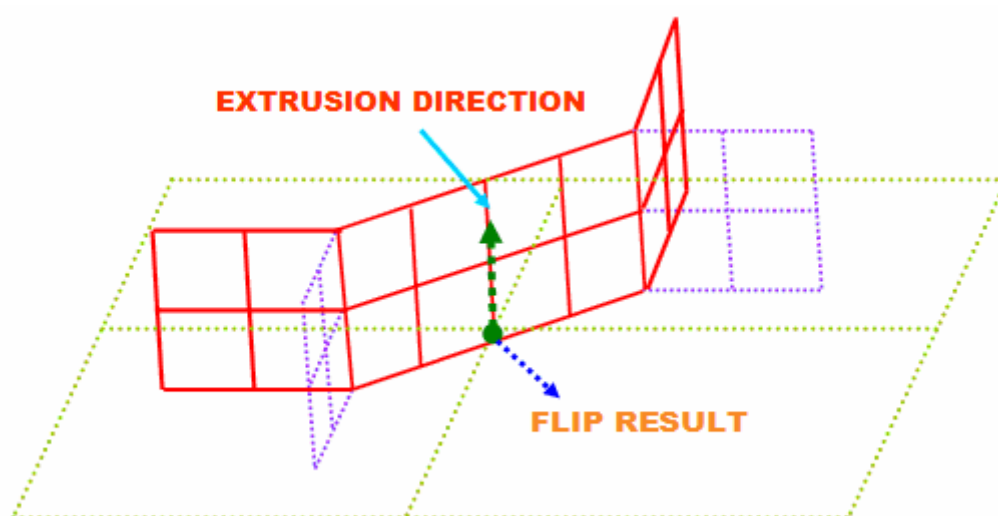


Figure 9.6.8 Flip result

**APPLY** – Creates the locator pin or guide.

**UNDO** – Undo the last operation.

**EXIT** – Exit the GUIDE & PIN dialog box.

## 9.7 DEFINE BLANK (CTRL+B)

This function allows the user to define the blank material and properties in the stamping simulation. The DEFINE BLANK dialog box is shown in Figure 9.7.1.

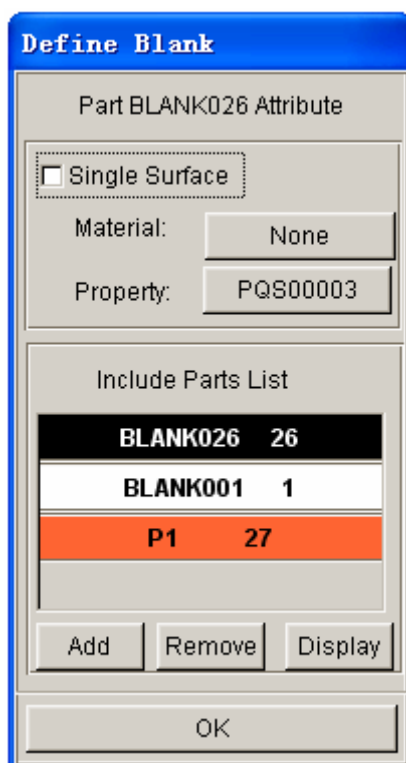


Figure 9.7.1 the Define Blank dialog box

### **SINGLE SURFACE**

This option allows the user to activate the single surface contact interface. Activating the single surface contact interface is recommended in order to accurately predict the double metal folding condition during stamping simulation.

### **ADD PART**

This function allows the user to add parts to the blank definition. It is similar to adding parts to tool definition (Section 9.1.1).

### **DEFINE BLANK MATERIAL**

This function allows the user to define the mechanical properties of the blank. Refer to Section 9.8.

### **DEFINE BLANK PROPERTY**

This function allows the user to define the shell element properties and the blank thickness. Refer to Section 9.9.

## 9.8 BLANK OPERATION

This menu contains eight different submenus which are used to manipulate the blank geometry, view deformed blank results, map blank results, etc. The submenus are listed below:

- BLANK AUTO POSITION
- BLANK MAPPING
- RESULT MAPPING
- DYNAMIC CONTOUR
- TRIM
- TIP
- TAILOR WELDED
- LANCING

### 9.8.1 BLANK AUTO POSITION

This function is aimed to automatically reposition the blank with reference to the selected tool and given contact gap. The blank can be positioned along X, Y or Z coordinates in either a global or local coordinate system. The dialog box is shown in Figure 9.8.1.

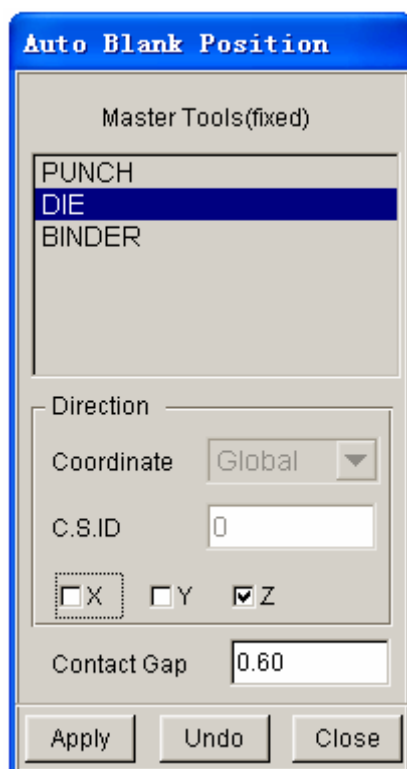


Figure 9.8.1 the Auto Blank Position dialog box

## 9.8.2 BLANK MAPPING

This function is utilized to project a flat blank with refined mesh onto a deformed blank with coarser mesh. This application is useful for converting the coarse blank mesh (from gravity loading simulation) to refined blank mesh needed for binder wrap and/or drawing simulation.

If the BLANK MAPPING is selected, the following message is printed in the message prompt window:

***SELECT COARSE MESH FOR THE FLAT BLANK***

The user selects the coarse blank mesh from the SELECT PART dialog box. The flat blank with coarse mesh is the initial blank mesh used in gravity loading simulation. After the selection of flat blank with coarse mesh, the following message is printed in the message prompt window:

***SELECT COARSE MESH FOR THE DEFORMED BLANK***

The user selects the deformed blank with coarse mesh that is produced by gravity loading simulation. After the selection of deformed blank with the coarse mesh from the SELECT PART dialog box, the following message is printed in the message prompt window:

***SELECT FINE MESH FOR THE FLAT BLANK***

The user selects the flat blank with the refined mesh. Both flat blanks share identical dimensions. eta/DYNAFORM will automatically map the refined blank mesh to the deformed blank shape. The deformed blank with finer mesh is then used for subsequent binder wrap and/or draw simulation.

## 9.8.3 RESULT MAPPING

The purpose of this function is two-fold:

- To map forming results to the car position part mesh for structural analysis.
- To tip a blank without interfering with the thickness or strain/stress information.

Before using the RESULT MAPPING function, the user must import the desired (targeted) part mesh in NASTRAN format into the eta/DYNAFORM database. The deformed blank (forming results) in DYNAIN format should then be read into the database. The user will continue to carry out the RESULT MAPPING operation using the dialog box as illustrated in Figure 9.8.2. If no DYNAIN file is read into the current database, the RESULT MAPPING function is disabled.

The procedure for the RESULT MAPPING operation is listed as following:

1. Click on both the FROM and TO buttons to select the deformed blank and the desired (targeted) part mesh, respectively.

2. Toggle on the COORDINATE TRANSFORM checkbox, following by defining (select buttons) or input (already existing) two coordinate system IDs to align the deformed part to the desired (targeted) part.
3. If the deformed blank is not aligned to the new blank, select the INCREMENTAL ADJUSTMENT option to adjust the position of the deformed blank. The user can input or define a C.S ID. Then, input the value and select the direction (U, V, W, RU, RV, RW). The user can click on the (+) and (-) buttons to manually adjust the deformed blank position. After aligning the two parts, select the MAP function to complete the RESULT MAPPING operation.
4. Select the RESET function to return the deformed part to its original position. Click on the EXIT button to dismiss the RESULT MAPPING dialog box without mapping the results.

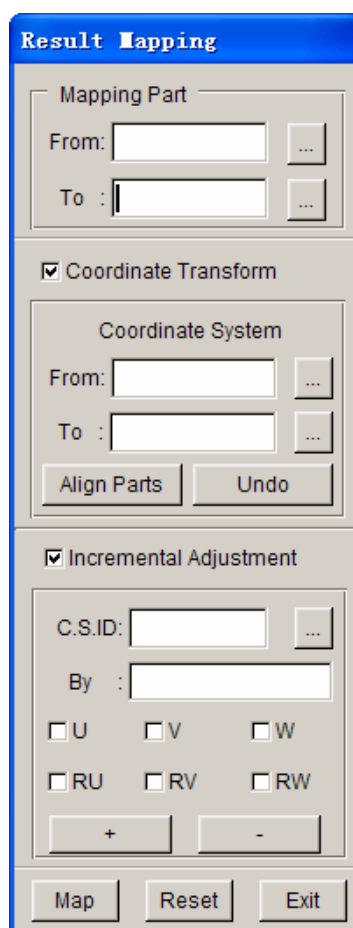


Figure 9.8.2 Result Mapping dialog box

#### 9.8.4 DYNAIN CONTOUR

This function allows the user to plot thickness contour, stress and strain distribution after the deformed blank (in DYNAIN format) is read into the eta/DYNAFORM database. If no blank result is available in

the eta/DYNAFORM database, this function is automatically disabled.

### 9.8.5 TRIM

This function is used to trim a blank using a closed curve trim line(s). The trimming operation is processed based on a defined vector/direction. The user can decide to keep the elements inside/outside of the trim line. These discarded elements are included in a new part called TRIMOUT.

After a trim line is selected, the user has to define a vector (direction of projection) for the trim curve, that is, W-direction or global Z-axis, from the LCS dialog box (as described in Section 2.9). The user can then set the trimming tolerance from the dialog box as shown in Figure 9.8.3. The trimming tolerance ranges between 0 and 1. It limits the size of the small elements generated during the trimming operation. A larger tolerance produces large elements, while a smaller tolerance generates small elements and more details. The default value is set to 0.30.

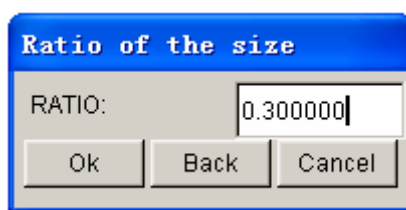


Figure 9.8.3 Tolerance Input dialog box

**NOTE:** *Prior to performing the trimming operation, the user must define the blank properties.*

### 9.8.6 TIP

This function allows the user to tip the blank according to a user-defined coordinate system. The dialog box is shown in Figure 9.8.4.

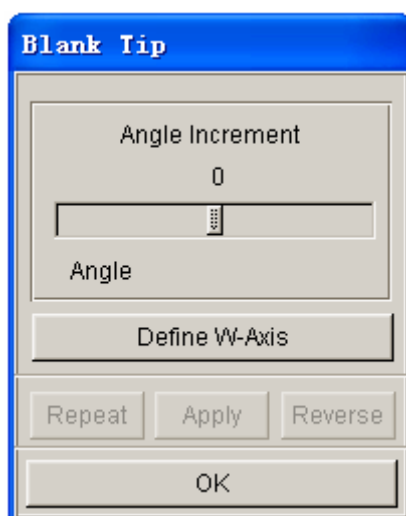


Figure 9.8.4 the Blank Tipping dialog box

As shown in Figure 9.8.4, the user can define the coordinate system by clicking on the DEFINE W-AXIS button. After a coordinate system is defined, the user can tip the blank using the REPEAT and REVERSE buttons. The slider bar is used to adjust the angle increment of tipping.

### 9.8.7 TAILOR WELDED

This function allows the user to weld together parts of different material and/or thickness. If the TAILOR WELDED function is selected, eta/DYNAFORM displays the SELECT PART dialog box, as shown in Figure 4.9.1, and prints the following message in the message prompt window:

***SELECT TARGET PART FOR TAILOR***

After the target part is selected, eta/DYNAFORM displays the SELECT NODE dialog box, as shown Figure 6.4.2, and prints the following message in the message prompt window:

***SELECT NODES TO TAILOR WELD***

After the nodes are chosen, the number of spot-weld nodes created by eta/DYNAFORM is listed in the message prompt window. The welded part is shown in the display area.

### 9.8.8 LANCING

This function allows user to carry out lancing operations by splitting the selected elements along a given path. If this function is selected, the user must select the elements using the SELECT ELEMENT dialog box. The SELECT METHOD dialog box, as shown in Figure 9.88.5, is displayed after the user



confirms the element selection.

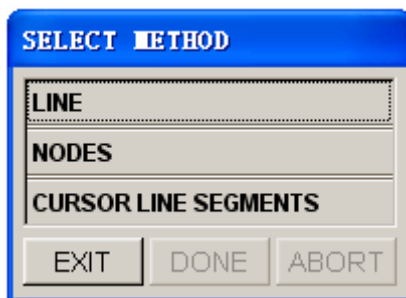


Figure 9.8.5 Select Method dialog box

### **LINE**

Allow the user to select an existing line as a splitting path.

### **NODES**

Allow the user to create a line segment by selecting the nodes. The line segment is projected on the selected elements along the defined W-AXIS. It is used as the splitting path.

### **CURSOR LINE SEGMENTS**

Generate the splitting path using line segments.

## **9.9 MATERIAL**

The MATERIAL dialog box, as illustrated in Figure 9.9.1, allows the user to create, modify, and delete material definition. In addition, the user can import and export any defined materials.

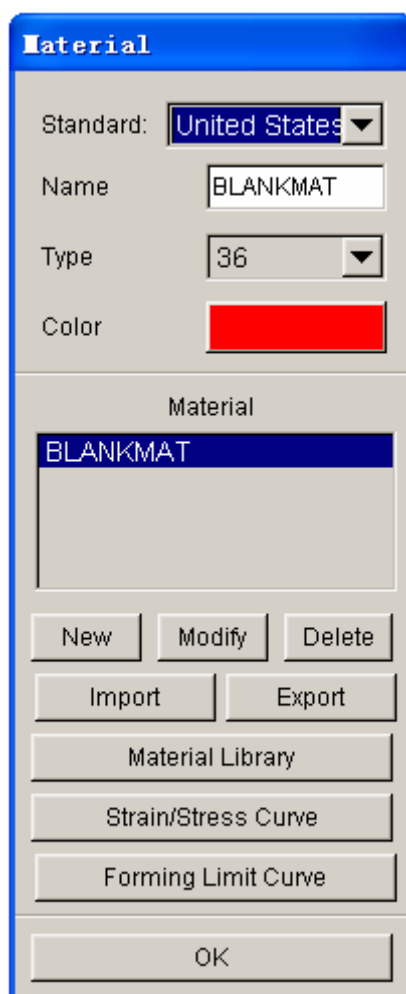


Figure 9.9.1 Material dialog box

The user can select the type of material model by clicking the drop down button as shown in Figure 9.9.1. The default material model is set to type #36. Refer to the LS-DYNA User's Manual for detailed information about each material model supported by eta/DYNAFORM.

### 9.9.1 NEW (MATERIAL)

This function enables the user to create a new material model, as well as defining the associated parameters in the table as illustrated in Figure 9.9.2.

MATERIAL TYPE 36	
MATERIAL TITLE	BLANKMAT
MASS DENSITY	7.830000E-009
YOUNGS MODULUS	2.070000E+005
POISSONS RATIO	2.800000E-001
HARDENING RULE(EXPON.)	2.000000E+000
MATERIAL PARAM P1 (K)	6.480000E+002
MATERIAL PARAM P2 (N)	2.200000E-001
EXPONENT FACE M	6.000000E+000
LANKFORD PARAM R00	1.870000E+000
LANKFORD PARAM R45	1.270000E+000
LANKFORD PARAM R90	2.170000E+000
FLD CURVE NUMBER	0

OK   Advanced   Default   Reset   Cancel

Figure 9.9.2 Material Model Parameter Table

**NOTE:** The FLD CURVE NUMBER is listed at the end of the every material table. This item is not needed by LS-DYNA except the MAT-39. This information is output to the INDEX file during the output of LS-DYNA input files. The information is utilized by eta/POST in the FLD option. The user can define the FLD curve for the material in the Forming Limited Curve button. Please refer to Section 9.9.8.

### 9.9.2 MODIFY (MATERIAL)

This function allows the user to edit the parameters of a selected material model using the table as shown Figure 9.9.2.

### 9.9.3 DELETE (MATERIAL)

This function is utilized to remove a defined material model from the database.

### 9.9.4 EXPORT

This function allows the user to save any defined material to a file with \*.mat extension. The EXPORT button displays the EXPORT MATERIAL window as shown in Figure 9.9.3.

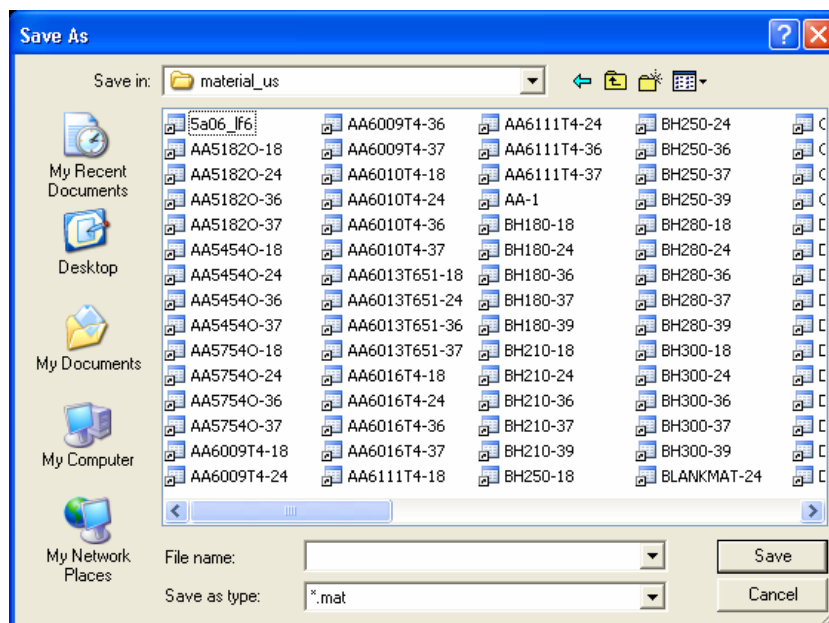


Figure 9.9.3 Export Material Window

### 9.9.5 IMPORT

This function allows the user to read in a material from a file with the \*.mat extension. Choose a material file and select OK to close the IMPORT MATERIAL window. The imported material is displayed in the MATERIAL window.

### 9.9.6 MATERIAL LIBRARY

This function allows user to select a material directly from the generic material library embedded in eta/DYNAFORM. In the eta/DYNAFORM 5.5, there are four types of material library: Europe, China, Japan and United States. The user can customize the default type of material library through the OPTION/MATERIAL LIBRARY menu. The material library is a folder located in the installation directory, “./installation directory/material\_lib/...”. It consists of \*.mat files and DF\_MATERIAL\_INDEX.dat file. Each \*.mat file consists of the LS-DYNA material model. The DF\_MATERIAL\_INDEX.dat is the index of all \*.mat files.

As shown in Figure 9.9.4, the user can select any material from the menu and the relative parameters are displayed in the MATERIAL PARAMETERS table.

Dynaform Material Library		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 9.9.4 Material Library window

**Disclaimer:** The user should take full responsibility for using the parameters in the eta/DYNAFORM material library. ETA will assume no responsibility for the accuracy or reliability of these generic data!

### 9.9.7 STRAIN/STRESS CURVE

The functions in this menu are used to create, read, and display stress/strain curves of materials. There are six options in this menu, as shown in Figure 9.9.5. The function of each option is similar to those described in Section 11.13.

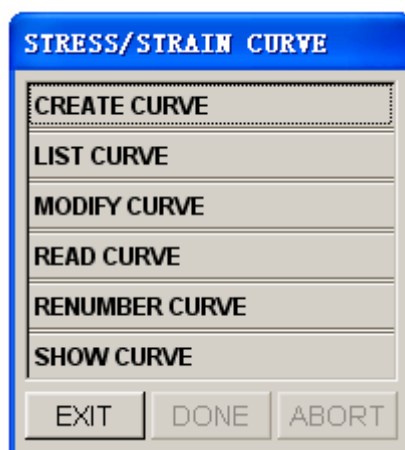


Figure 9.9.5 Stress/Strain Curve dialog box

### 9.9.8 FORMING LIMIT CURVE

The functions in this menu are used to create, read, and display the FLD curve of the materials. The dialog box is shown in Figure 9.9.6. These options are similar to those described in Section 11.13.

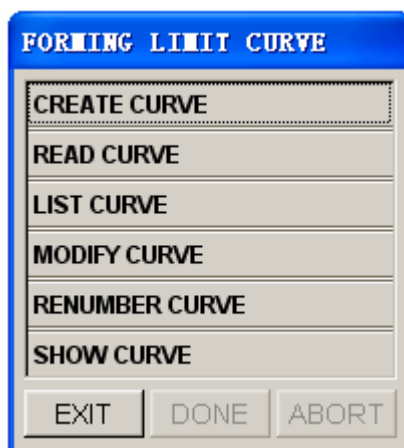


Figure 9.9.6 Forming Limit Curve dialog box

#### CREATE CURVE

This option enables the user to create a FLC Curve based on the Keeler's empirical formula. There are two methods to define the FLD0. One is by calculating the FLD0 through the thickness (t) and hardening parameter (n), while the other one is by inputting the FLD0 directly. Figure 9.9.7 illustrates the CREATE FLC dialog box.

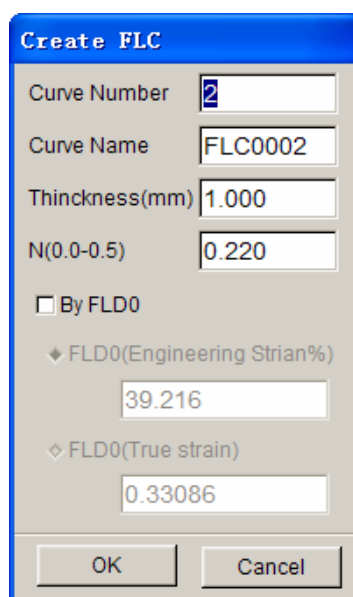


Figure 9.9.7 Create FLC dialog box

**CURVE NUMBER** – Enables the user to define the curve ID for the current FLC curve.

**CURVE NAME** - Enables the user to define the curve name for the current FLD curve.

**THICKNESS (MM)** - Enables the user to define the thickness required by the Keeler's empirical formula for calculating the FLD0. The unit of the thickness is in millimeter.

**HARDENING EXPONENT (N)** - Enables the user to define the hardening exponent (n) required by the Keeler's empirical formula for calculating the FLD0.

**BY FLD0** – This function enables the user to manually input the user-defined FLD0. The default FLD0 is calculated by the Keeler's empirical formula. However, the user can edit the value. Usually, the FLD0 and FLC are expressed in engineering strain. The true strain FLD0 and FLC is also supported.

**NOTE:** *There are three different curve formats that are supported by eta/DYNAFORM; These curve formats can be read into an eta/DYNAFORM database.*

**FORMAT 1:** FEMB read in format

```
$ THIS IS THE FEMB FORMAT LOAD CURVE
$ FEMB LOAD CURVE (TITLE LINE HAVE BEEN EXIST)
$CURVE, CURVE ID, TYPE, CURVE NAME (A9, I9, I9, 1X, A8)
CURVE 9 0 CURVE9
$CURVE DATA (X, Y) (F9.0, F9.0)
0.0000E+000.1349E+03
0.900E-09.2018E+03
0.3000E-09.269E+03
0.9000E-09.3017E+03
0.7000E-09.3282E+03
0.9000E-09.3494E+03
0.190E+000.3674E+03
0.1300E+000.3831E+03
0.1900E+000.3970E+03
0.1700E+000.4097E+03
0.1900E+000.4212E+03
0.290E+000.43199E+03
0.2300E+000.4418E+03
0.2900E+000.4911E+03
0.2700E+000.4999E+03
0.2900E+000.4682E+03
```

```
0.9990E+09.4722E+02
```

```
END
```

**FORMAT 2:** FEMB write out format (can be read in also)

```
1 (CURVE NUMBER, I9)
  27 'fld' 90 (num. of points, 'curve name', curve id) (4x, i, 'a8', i4)
-4.94296E-01  9.00160E-01
-4.46287E-01  8.92777E-01
-4.00478E-01  8.06437E-01
-3.96679E-01  7.61400E-01
-3.14711E-01  7.17994E-01
-2.74437E-01  6.76406E-01
-2.39722E-01  6.37089E-01
-1.98491E-01  6.00334E-01
-1.62919E-01  9.66903E-01
-1.27833E-01  9.39946E-01
-9.43970E-02  9.09004E-01
-6.18794E-02  4.86003E-01
-3.04992E-02  4.67236E-01
 0.00000E+00  4.92994E-01
 2.99988E-02  4.71172E-01
 9.82689E-02  4.86212E-01
 8.61777E-02  4.98662E-01
 1.13329E-01  9.08994E-01
 1.39762E-01  9.17989E-01
 1.69914E-01  9.24738E-01
 1.90620E-01  9.30703E-01
 2.19111E-01  9.39683E-01
 2.39017E-01  9.39843E-01
 2.62364E-01  9.43322E-01
 2.89179E-01  9.46232E-01
 3.07489E-01  9.48669E-01
 3.29304E-01  9.90709E-01
```

**FORMAT 3:** (DYNA format)

```
*KEYWORD (must)
```

```
*DEFINE_CURVE
```

```
$CURVENAME      ABC
$   LCID      SIDR      SCLA      SCLO      OFFA      OFFO
      200        0
$           A1           01
      .000000000E+00      .000000000E+00
      .476209790E+00      .770189471E+02
```



```

.992419979E+00      .122399673E+03
.142862940E+01      .149092189E+03
.190483916E+01      .164766199E+03
.238948920E+01      .174017044E+03
.289729880E+01      .179463074E+03
.333346844E+01      .182669189E+03
.380967832E+01      .184996629E+03
.428988820E+01      .189667770E+03
.476209784E+01      .186321899E+03

```

\*END

(Here the \*END is optional)

## 9.10 PROPERTY

The functions in this menu are used to define and modify the physical properties of the blank. As shown in Figure 9.10.1, the PROPERTY dialog box allows the user to define the type of shell element formulation for the stamping simulation. The default shell element formulation is BELYTSCHKO-TSAY, which is widely implemented in stamping simulation. Please refer to the LS-DYNA User's Manual for detailed information about the shell element formulation.

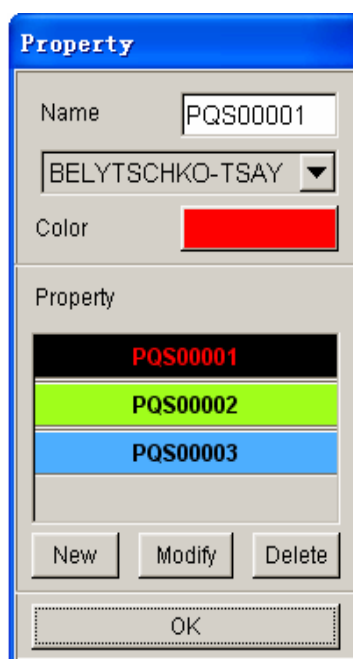


Figure9.10.1 Property dialog box

### 9.10.1 NEW

This function allows the user to create a new blank property. The user can define the name of the new

blank property, thickness and through thickness integration points from the PROPERTY table.

The functions MODIFY and DELETE, are similar to those described in Section 9.9.

## 9.11 ANIMATE

This function is used to show the motion of all defined tools with either velocity or a displacement load curve. The user will not be able to view the motion of tools with a force load curve. The animation dialog window is shown in Figure 9.11.1.

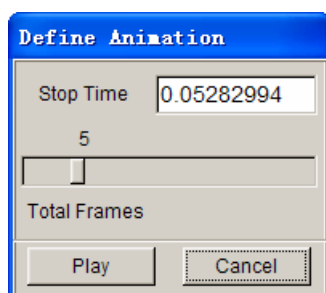


Figure 9.11.1 Define Animation dialog box

### STOP TIME

The default stop time is equal to the termination time of the simulation. The user can also set a stop time that is less than the termination time.

### SLIDER

Allow the user to set the number of frames for the animation.

### PLAY

Allow the user to animate all frames. It also displays the ANIMATE dialog box as shown in Figure 9.11.2.

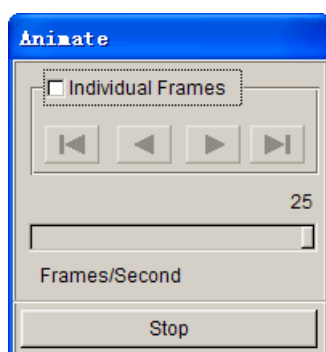


Figure 9.11.2 Animate dialog box

As shown in Figure 9.11.2, the user can animate the tool motion using the INDIVIDUAL FRAMES

option. When the function is toggled on, the user can select the FORWARD and REVERSE buttons to view the animation. The slider bar is used to adjust the speed of animation.

## 9.12 TOOLS ON/OFF

This function allows the user to toggle on/off individual or all tools. The TOOLS ON/OFF dialog box is shown in Figure 9.12.1.

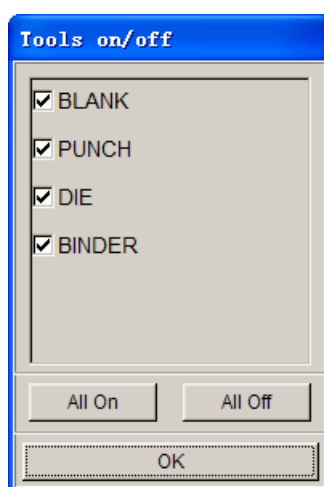


Figure 9.12.1 Tools On/Off dialog box

## 9.13 SUMMARY

This function enables the user to review the statistical information of each tool from the SELECT TOOLS dialog box.

After a tool is selected, a message window with tool statistics is displayed. An example of the message window is shown in Figure 9.13.1.

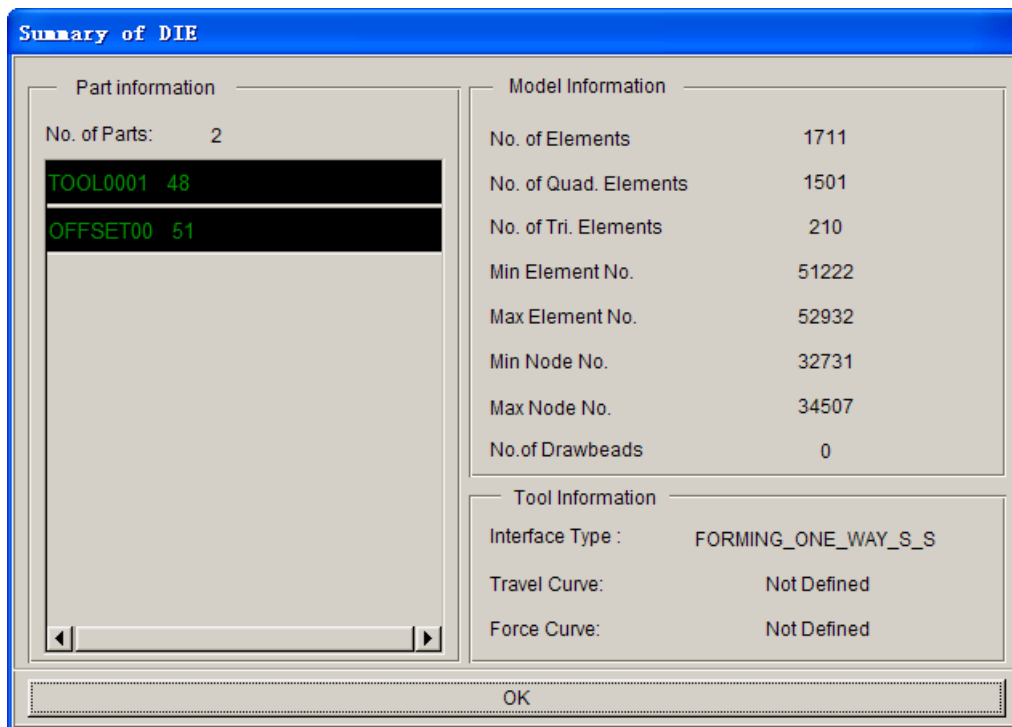


Figure 9.13.1 Tool Statistic Message window

## Chapter 10

# OPTION MENU

This menu consists of various preprocessing utilities that are toggle-switch activated as shown in Figure 010.1.

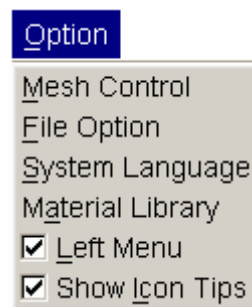


Figure 10.1 Option Menu

A detailed description of these functions is given in the following sections.

## 10.1 MESH CONTROL

The options in this dialog box, as illustrated in Figure 10.1.1, are used to control mesh generation.

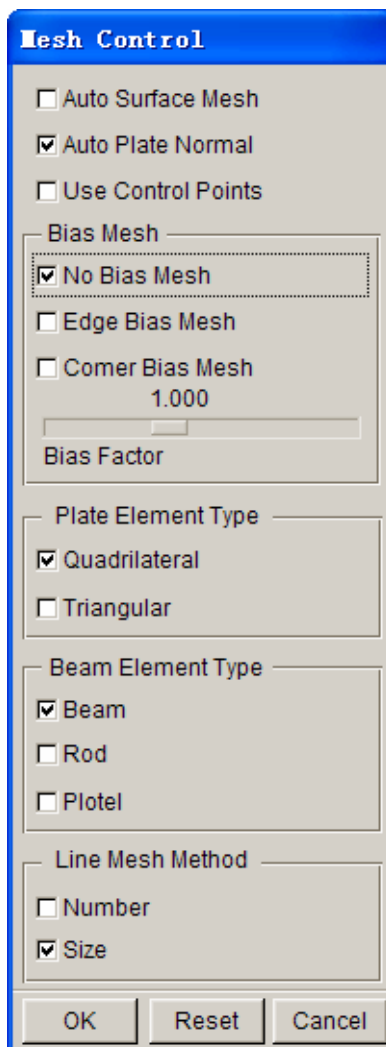


Figure 10.1.1 Mesh Control dialog box

### **AUTO SURFACE MESH (toggle)**

This function enables the user to mesh the displayed surfaces directly using the SURFACE MESH function.

### **AUTO PLATE NORMAL (toggle)**

This function enables the user to mesh the part with consistent element normal.

### **10.1.1 CONTROL POINT (toggle)**

This function enables the user to pre-select the points on a line for the desired node locations associated with the 2L, 3L, 4L, 6S, 10S, 9S, and 12S mesh function. It is used in conjunction with the options in Section 5.3.4.

### 10.1.2 EDGE BIAS MESH

The EDGE BIAS MESH command allows the user to enter an element bias factor (0.625 to 1.6) relative to the selected edge of the line data that is to be modeled. This function multiplies the size of each adjacent element that is created from the selected edge by the bias factor.

If the EDGE BIAS MESH function as shown in Figure 10.1.1 is toggled on, the user can define the bias factor by dragging the slider. An example of mesh with edge bias mesh control is shown in Figure 10.1.2b.

**NOTE:** A bias factor greater than 1.0 generates proportionately larger elements from the selected edge. A bias factor less than 1.0 generates proportionately smaller elements from the selected edge. This function can only be implemented with the 2L, 3L, 4L, 6S, 10S, 9S, and 12S meshing function.

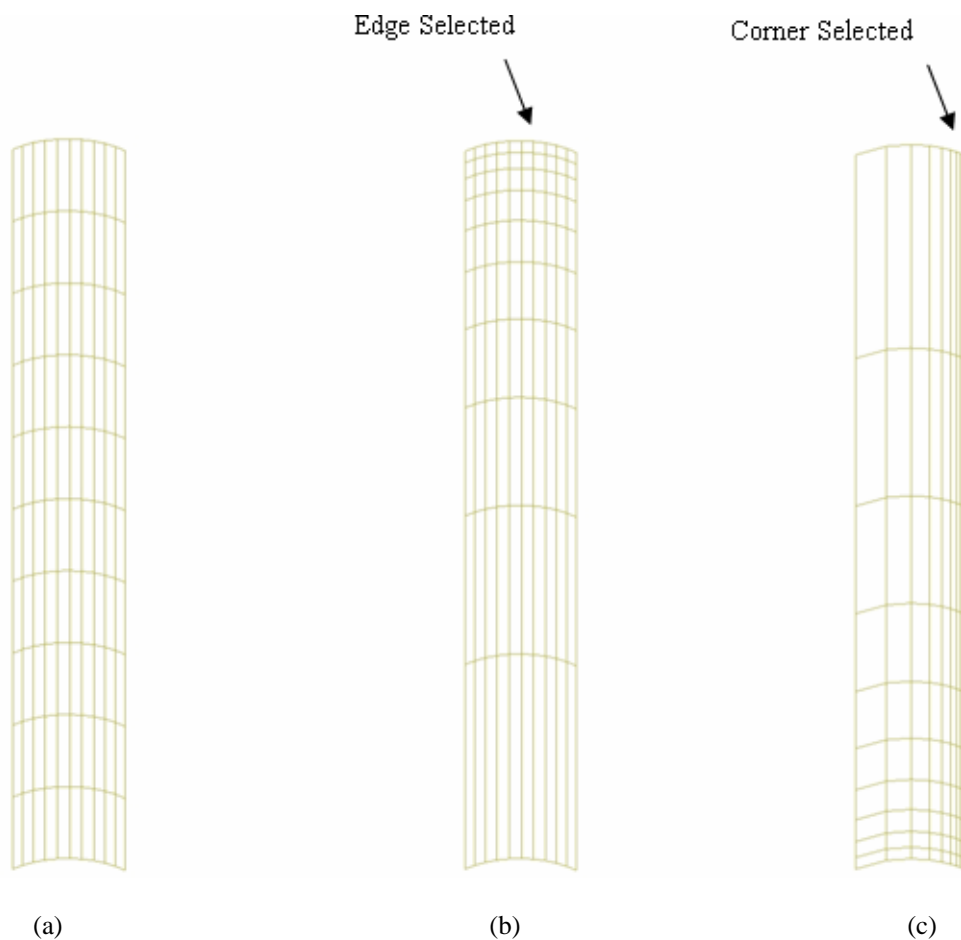


Figure 10.1.2 An example of mesh with bias mesh control: (a) No Bias (b) Edge Bias (c) Corner Bias (Bias Factor=1.765).

### 10.1.3 CORNER BIAS MESH

The CORNER BIAS MESH control allows the user to enter an element bias factor (0.625 to 1.6) relative to the selected corner of the line data that is to be modeled. This function multiplies the size of each adjacent element that is created from the selected corner by the bias factor.

If the CORNER BIAS MESH function as shown in Figure 10.1.1 is toggled on, the user can define the bias factor by dragging the slider. An example of mesh with corner bias mesh control is shown in Figure 10.1.2c.

**NOTE:** *A bias factor greater than 1.0 generates proportionately larger elements from the selected corner. A bias factor less than 1.0 generates proportionately smaller elements from the selected corner. This function can only be implemented with the 2L, 3L, 4L, 6S, 10S, 9S, and 12S meshing function.*

### 10.1.4 PLATE ELEMENT TYPE

This function enables the user to control the type of mesh generated by eta/DYNAFORM. If the QUADRILATERAL function as shown in Figure 10.1.1 is toggled on, the plate mesh generated is quadrilateral element dominant with minimum triangular elements. This captures the tool geometry accurately and efficiently, including the outer edge of the irregular blank shape. If TRIANGLE option is checked, the mesh generated is triangular element dominant.

### 10.1.5 LINE MESH METHOD

Please refer to Section 6.3.5 for detailed information about LINE MESH METHOD.

## 10.2 FILE OPTION

This function enables the user to set the FILE DIALOG STYLE (WINDOWS OR UNIX) and set the AUTO BACKUP file saving function. The dialog box is shown in Figure 10.2.1.

**NOTE:** *The FILE DIALOG STYLE option is inactive in WINDOW.*



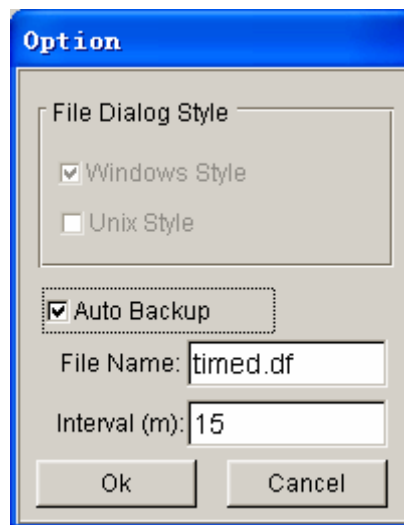


Figure 10.2.1 Option dialog box

If WINDOWS STYLE is checked, the file dialog style is shown as in Figure 10.2.2

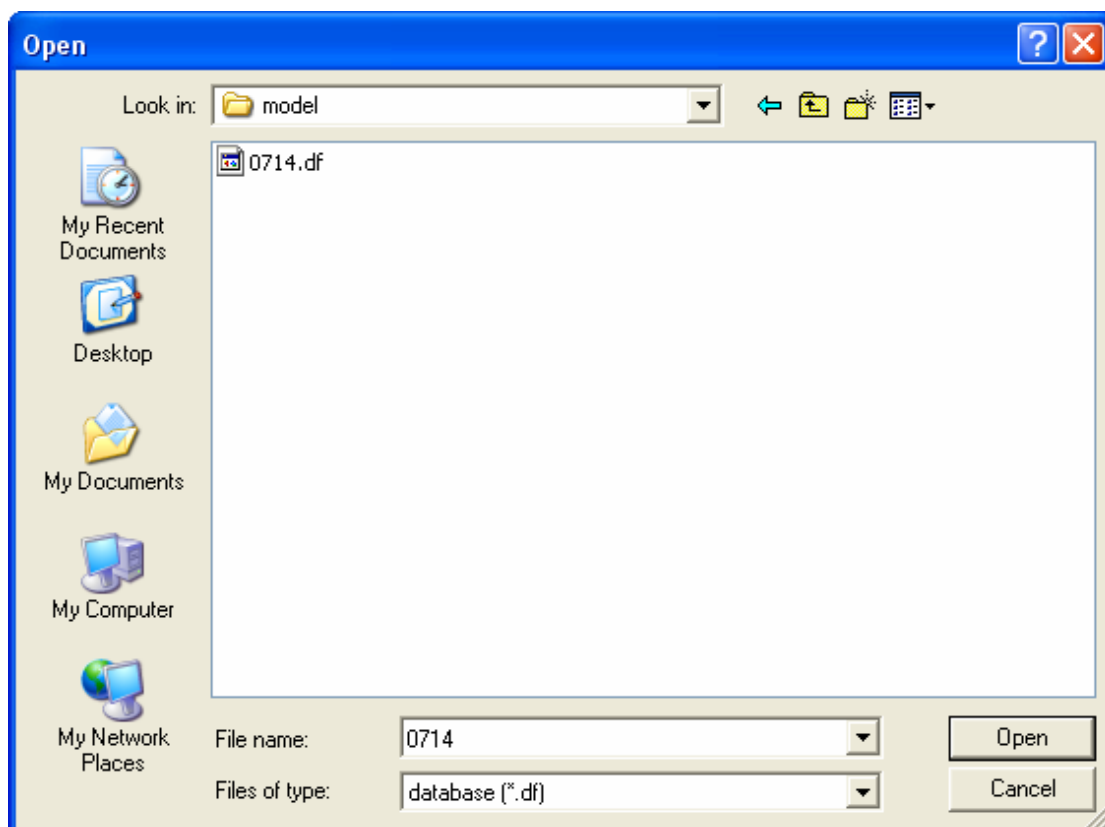


Figure 10.2.2 Windows Dialog Style

If UNIX STYLE is checked, the file dialog style is shown as in Figure 10.2.3.

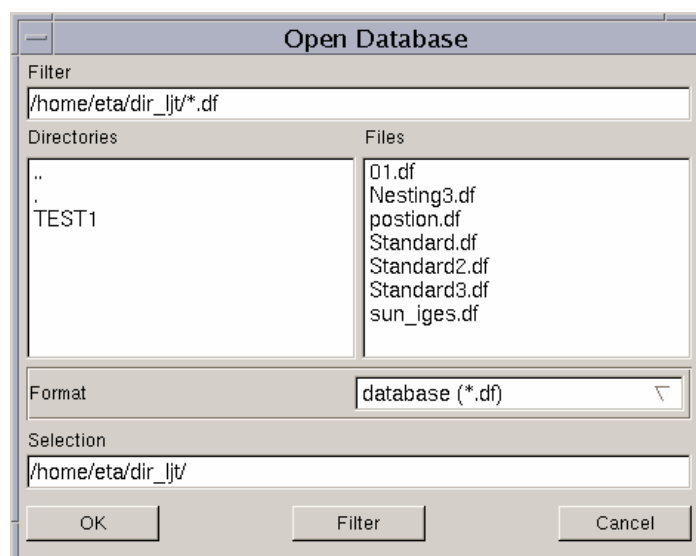


Figure 10.2.3 UNIX Dialog Style

### 10.3 SYSTEM LANGUAGE

This function allows the user to change the language in the menu, prompt and icon tips displayed on the screen. The program displays a dialog box as shown in Figure 10.3.1.

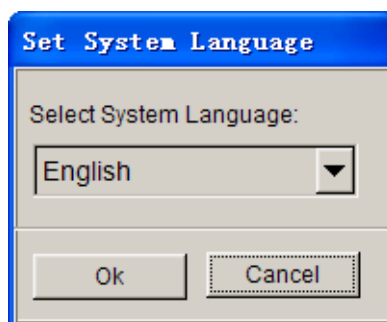


Figure 10.3.1 Select Language dialog box

Click on the drop down button to select the language. Currently eta/DYNAFORM supports four languages: Chinese, English, Japanese and Korean. More languages can be customized for eta/DYNAFORM using a resource compiler utility software.

After a language is selected, an eta/DYNAFORM question window as shown in Figure 10.3.2 is displayed.

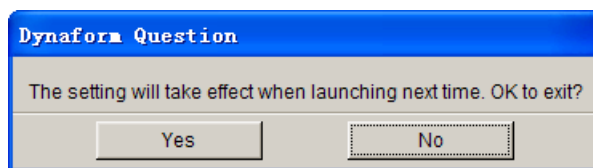


Figure 10.3.2 Select Language Confirmation Window

If YES is selected, eta/DYNAFORM sets the selected language as the default language in all future eta/DYNAFORM sessions. Select NO to keep the current language.

**NOTE:** The user may also use the *SYSTEM LANGUAGE* setting in the configuration file *dynaform.ini* to set the default language.

## 10.4 MATERIAL LIBRARY STANDARD

There are four different types of material library supported in eta/DYNAFORM10.4. The user can set the default material library from the SET STANDARD dialog box as illustrated in Figure 10.4.1. The user also can switch the type of material library from the MATERIAL dialog box. Please refer to the Section 9.9 for information about the Material.

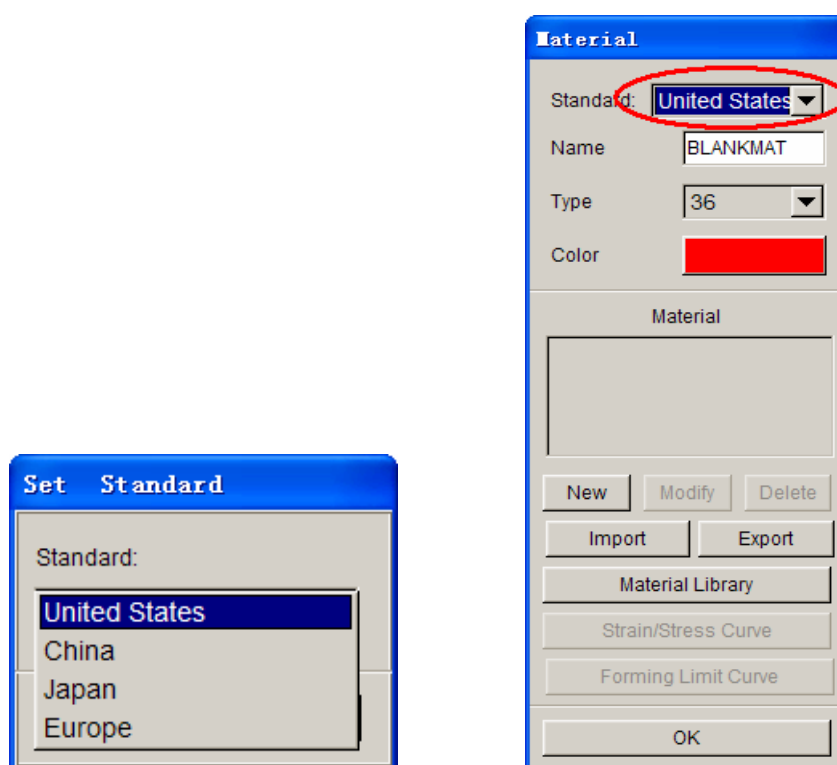


Figure 10.4.1 Set the default material standard

## 10.5 LEFT MENU

The pop-up menu is default located on the left of the mainly GUI of the eta/DYNAFORM. User can change the style and make the pop-up menu located on the right of the mainly GUI as the Figure 10.5.1 shown.

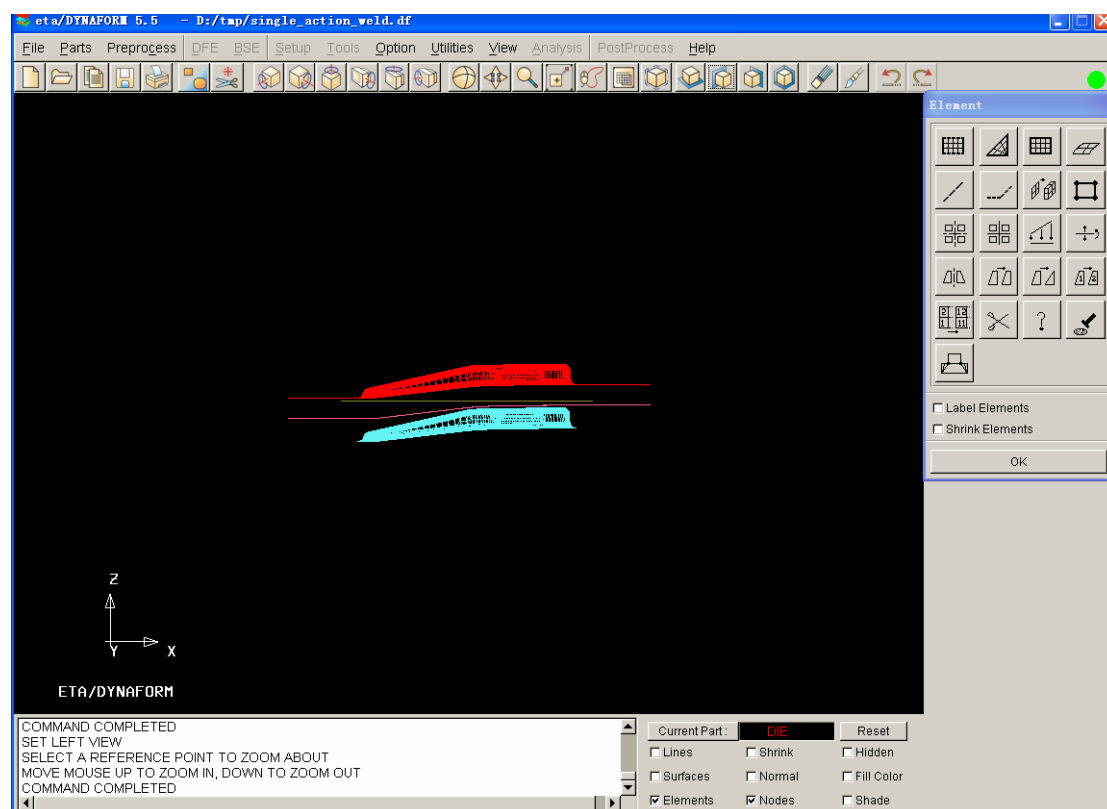


Figure 10.5.1 Left menu option

## 10.6 SHOW ICON TIPS (toggle)

This function allows the user to read/identify the name of each icon on the menu bar and dialog box. When the user places the cursor over the icon for a short moment, the tips are shown on the bottom-right of the icon such as illustrated in Figure 10.6.1.

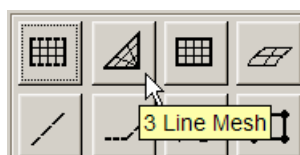


Figure 10.6.1 Show Tips

## Chapter 11

# UTILITIES

The functions in the UTILITIES menu make up eta/DYNAFORM's tool kit. Many of these functions are also found in other menus, but the UTILITIES menu (as shown in Figure 11.1) provides the user with a convenient way to access these functions.

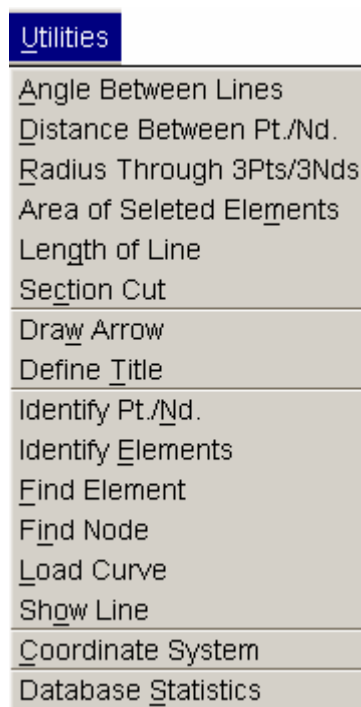


Figure 11.1 Utilities Menu

A detailed description of each function is given in the following sections.

## 11.1 ANGLE BETWEEN LINES

This function enables the user to measure the angle (in degrees) between two selected vectors.

After selecting the function, the INPUT COORDINATE dialog box is displayed. The user needs to define two vectors by selecting 3 points/nodes. eta/DYNAFORM displays the measurement of four angles in the message prompt window in the following format:

*ANGLE=69.90, TOP: 72.49, SIDE: 3.41, REAR: 124.78*

## 11.2 DISTANCE BETWEEN POINTS/NODES

This function enables the user to measure the distance (in mm) between two points/nodes, or a node and a point. This function is also provided in the PREPROCESS/NODE dialog box.

The user can select two points/nodes by using options provided in the INPUT COORDINATE dialog box. The distance and the differences of the three components are printed in the message prompt window in the following format:

*D=180.000, DX=180.000, DY=0.000, DX=0.000*

## 11.3 RADIUS THROUGH 3PTS/3NDS

This function is used to measure the radius (in mm) using 3 points/nodes.

The user can select three points/nodes by using options provided in the INPUT COORDINATE dialog box. The result is printed in the message prompt window in the following format:

*R=20.000 CENTER AT, X=-200.000, Y=0.000, Z=0.000*

## 11.4 AREA OF SELECTED ELEMENTS

This function is used to calculate the area of the selected elements. The area and mass center are printed in the message prompt window in the following format:

*AREA: 1000.0000, MASS CENTER: 1000.00000, 0.00000, 0.00000*

## 11.5 LENGTH OF LINE

This function allows user to measure the length of the selected line. After selecting this function, the SELECT LINE dialog box as illustrated in Figure 11.5.1 is displayed.

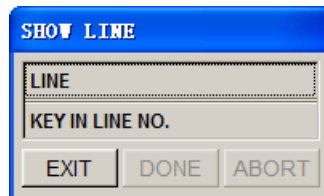


Figure 11.5.1 Select Line dialog box

If the user selects the LINE option, eta/DYNAFORM prompts the user to select a line on the display area. If the user selects the KEY IN LINE NO. option, the LINE NUMBER dialog box as illustrated in Figure 11.5.2 is displayed. The user can select the line through inputting the line ID.

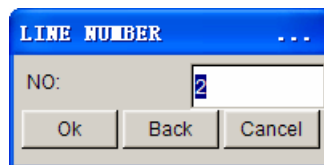


Figure 11.5.2 Line Number dialog box

The length of the selected line is printed in the message prompt window in the following format:

***LINE SEGMENT LENGTH IS: 232.3619***

## 11.6 SECTION CUT

This function allows the user to create one or several section lines, measure section length, measure angle and display section line. After selecting the function, the program will pop up a dialog box as show Figure 11.6.1.

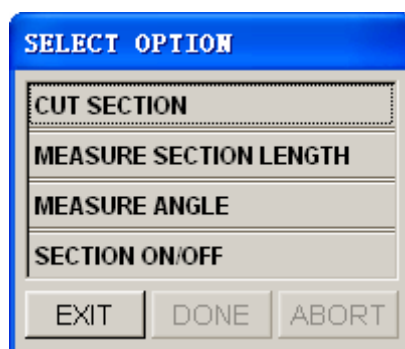


Figure 11.6.1 Section line option

### 11.6.1 CUT SECTION

This function allows the user to create one or several section lines. New section line is named SECTION in the part layer. After selecting the function, the program will pop up coordinate system dialog box. If the user selects the default Globe coordinate system, the section line created is perpendicular to Z axis. If the user creates Local coordinate system, created section line is perpendicular to W axis. After the coordinate system is created, the program will pop up the dialogue box as show Figure 11.6.2.

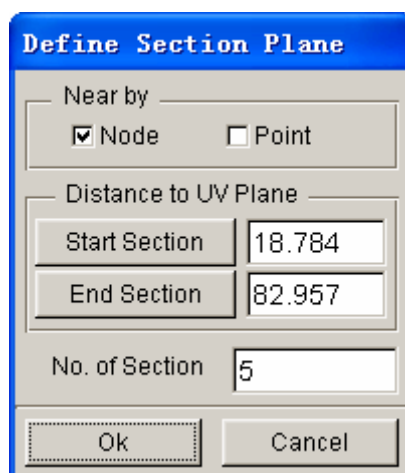


Figure 11.6.2 Define section plane

**NEAR BY:** Select the manner of picked point: node or point.

**START SECTION:** Select start point of section. Display the distance between the start point and U-V plane.

**END SECTION:** Select end point of section. Display the distance between the end point and U-V plane.

**NO.OF SECTON:** Numbers of section line between start section and end section.



### 11.6.2 MEASURE SECTION LENGTH

This function allows the user to measure section line length. After selecting the function, the program will pop up the dialog box as show Figure 11.6.3.

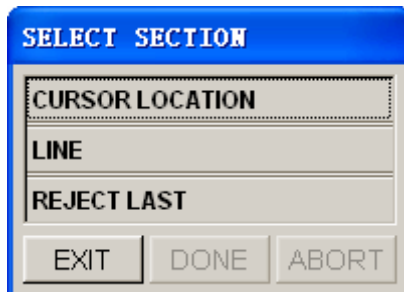


Figure 11.6.3 Select section dialogue window

**CURSOR LOCATION:**

Select two points of the section line by cursor, and then the length of straight line segment will be displayed in the message prompt area.

**LINE:**

Select a section line by cursor, and then the length of section line will be displayed in the message prompt area.

**REJECT LAST:**

Select the point through CURSOR LOCATION function, the first selected point can be canceled by clicking this option.

### 11.6.3 MEASURE ANGLE

This function allows the user to measure angle between two lines. The line is generated by two selected points. Measured angle information is displayed in the message prompt area.

### 11.6.4 SECTION ON/OFF

This function allows the user to only display SECTION layer on the screen, and set SECTION layer to current part layer.

## 11.7 DRAW ARROW

This function allows the user to draw arrows near a specific location on the display area.

The user selects two locations on the display area. An arrow is drawn from the first location to the second location.

**NOTE:** This function is used in conjunction with **DEFINE TITLE**. The arrow and the title can be removed using the **CLEAR HIGHLIGHT** icon.

## 11.8 DEFINE TITLE

This function allows the user to enter a title or text at any location of the displayed area. The dialog box is shown in Figure 11.8.1.

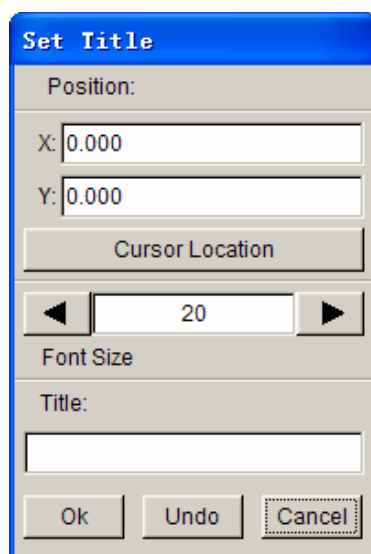


Figure 11.8.1 the Define Title dialog box

- Type in the caption in the Title input field.
- Click the arrow buttons to set the font size of the title.
- Click OK to display the title on the display area.

The user can define the position of the title by entering the screen coordinates of the X, Y values or by selecting **CURSOR LOCATION**.

## 11.9 IDENTIFY NODE/POINT

This function enables the user to identify any node/point identification number and its corresponding global location in the X, Y, Z coordinates. This function is also found in the **PREPROCESS/NODE** dialog box.

## 11.10 IDENTIFY ELEMENT

This function allows the user to identify an element's identification number and its associated nodes' identification number. It is also found in the PREPROCESS/ELEMENT dialog box.

## 11.11 FIND ELEMENT

This function allows the user to find or identify an element by inputting its element identification number. This function is also found in the PREPROCESS/ELEMENT dialog box.

## 11.12 FIND NODE

This function allows the user to find the X, Y, Z coordinates of the specified node by inputting the node identification number. This function is also found in the PREPROCESS/NODE dialog box.

## 11.13 LOAD CURVE

The options in this menu are utilized to generate or modify the load curves. The options are shown in Figure 11.13.1.

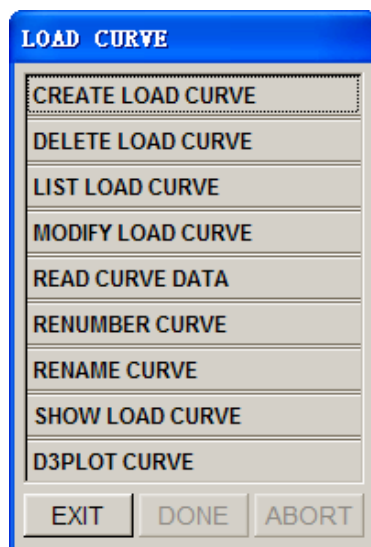


Figure 11.13.1 Load Curve Options

### 11.13.1 CREATE CURVE

This function allows the user to manually create a load curve. As shown in Figure 11.13.2, the user can type in both the curve number and name in the CREATE CURVE dialog box. A warning message is printed if the curve number is being used by another load curve or only one point is listed in the curve. The user must click on the ADD POINT button and key in the curve data manually.

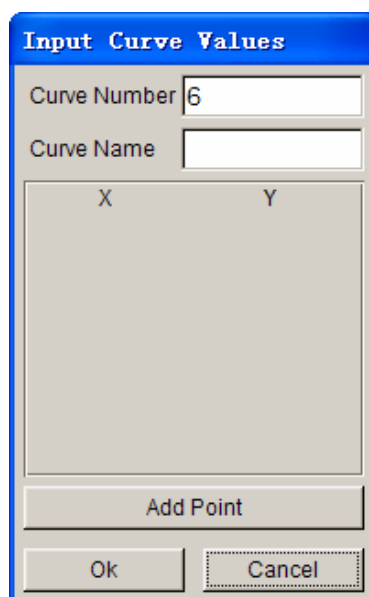


Figure 11.13.2 the Create Curve dialog box

### 11.13.2 DELETE LOAD CURVE

This function allows the user to delete an existing load curve from the database.

- Select a curve listed in the SELECT CURVE dialog box as shown in Figure 11.13.3. The user can select all curves by selecting the ALL button. The user can select several curves by holding down the CTRL or SHIFT key following by clicking on the OK button to confirm the selection.

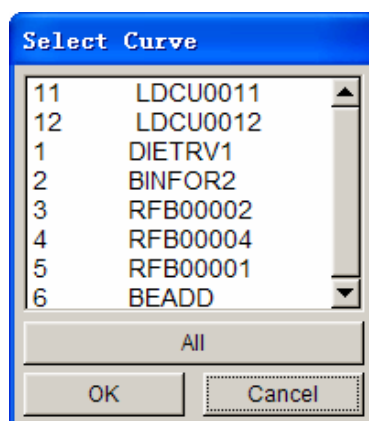


Figure 11.13.3 Select Curve dialog box

- An eta/DYNAFORM QUESTION dialog box, as shown in Figure 11.13.4, is displayed.

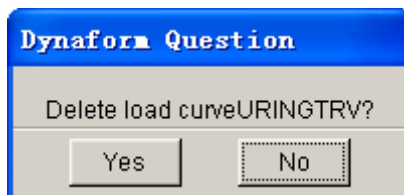


Figure 11.13.4 eta/DYNAFORM Question dialog box

- Select YES to delete the selected curve(s).
- Select NO to reject the choice.

### 11.13.3 LIST LOAD CURVE

This function allows the user to list existing load curves in the eta/DYNAFORM database. The user can select one or multiple load curves from the LIST CURVE dialog box as shown in Figure 11.13.5.

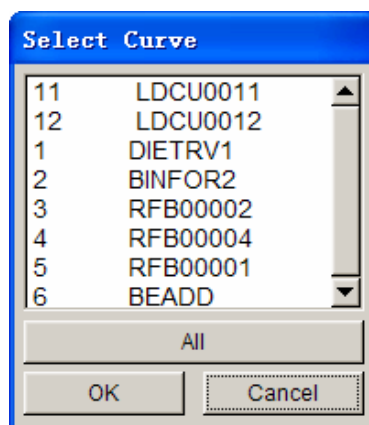


Figure 11.13.5 List Curve dialog box

### 11.13.4 MODIFY LOAD CURVE

This function allows the user to modify the data points of a selected load curve.

After selecting a curve in the SELECT CURVE dialog box, the SELECT OPTION dialog box, as illustrated in Figure 11.13.6, is displayed.

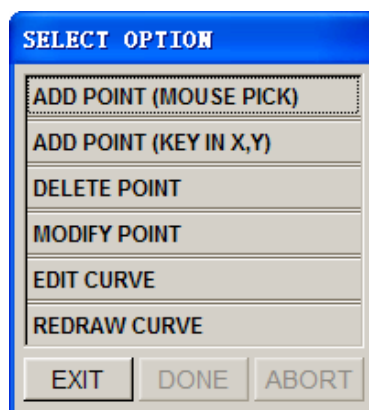


Figure 11.13.6 The Modify Curve dialog box

#### **ADD POINT (MOUSE PICK)**

Allow the user to select any location from the graph. The selected point is included in the curve.

#### **ADD POINT (KEY IN X, Y)**

Allow the user to add curve data points by keying in the X and Y values.

#### **DELETE POINT**

Removes selected data points.

#### **MODIFY POINT**

There are several options to modify data points using the SELECT OPTION dialog box as shown in Figure 11.13.7.

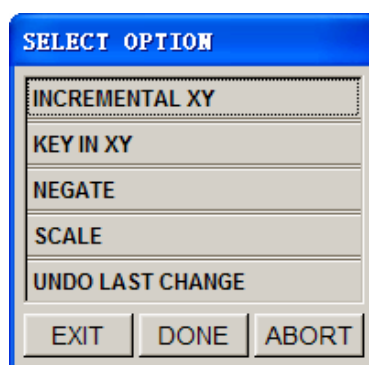


Figure 11.13.7 Modify Point Options

- **INCREMENTAL XY**  
Enter the X, Y increments, and select OK in the next dialog box. Select point(s)

to move in the curve window.

- **KEY IN XY**  
Select a point in the curve window. The X, Y values will be shown in the next dialog box. Edit the X, Y values, and select OK. The points on the curve is moved to the new coordinates. Repeat to modify another point.
- **NEGATE**  
Select point(s). The selected point(s) will be flipped about the X-axis.
- **SCALE**  
Enter the scale factors, and select OK in the next dialog box. Select point(s). The selected point(s) will move to the scaled coordinates.
- **UNDO LAST CHANGE**  
Reject the last modification.

### EDIT CURVE

This function enables user to modify the X, Y values of the curve. The EDIT CURVE dialog box is shown in Figure 11.13.8.

X	Y
0	0
0.26	1.3302
0.52	1.69018
0.78	1.70856
1.04	1.73918
1.3	1.78443

Figure 11.13.8 Edit Curve dialog box

## REDRAW CURVE

This function redraws the modified curve in the display area.

### 11.13.5 READ CURVE DATA

This function enables the user to import an existing curve's data into the eta/DYNAFORM database.

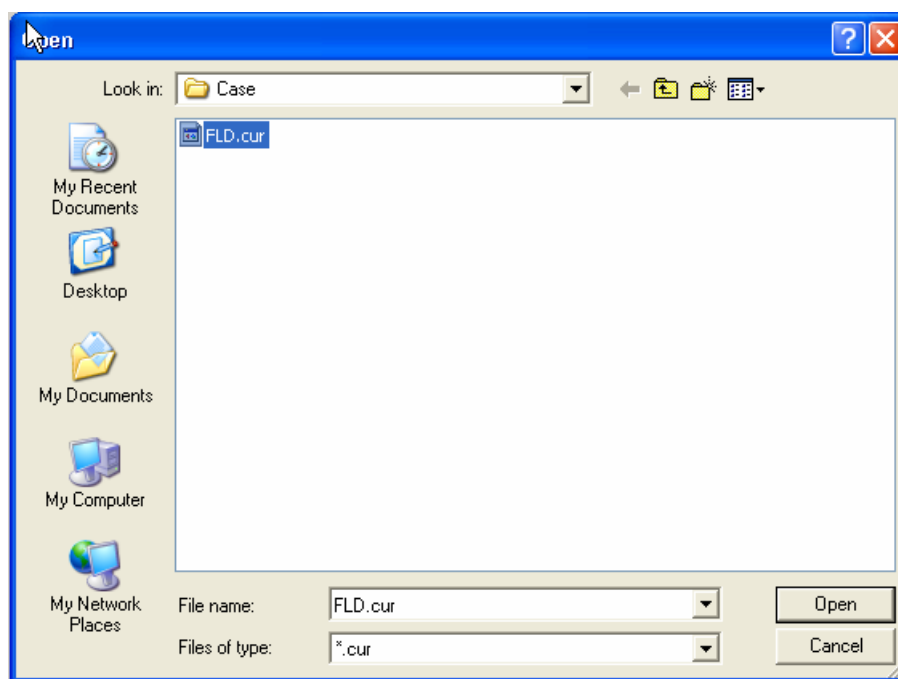


Figure 11.13.9 the Open Curve File Window

**NOTE:** *There are three different curve formats supported by eta/DYNAFORM; i.e., all three curve formats can be imported into eta/DYNAFORM.*

#### FORMAT 1: FEMB read in format

```
$ THIS IS THE FEMB FORMAT LOAD CURVE
$ FEMB LOAD CURVE (TITLE LINE HAVE BEEN EXIST)
$CURVE, CURVE ID, TYPE, CURVE NAME (A5,I5,I5,1X,A8)
CURVE 10 0 CURVE10
$CURVE DATA (X,Y) (F10.0,F10.0)
0.0000E+00,0.13411E+03
0.1000E-01,0.2018E+03
0.3000E-01,0.2655E+03
```



```

0.5000E-01,0.3017E+03
0.7000E-01,0.3282E+03
0.11000E-01,0.34114E+03
0.1100E+00,0.3674E+03
0.1300E+00,0.3831E+03
0.1500E+00,0.31170E+03
0.1700E+00,0.40117E+03
0.11100E+00,0.4212E+03
0.2100E+00,0.43111E+03
0.2300E+00,0.4418E+03
0.2500E+00,0.4511E+03
0.2700E+00,0.451111E+03
0.21100E+00,0.4682E+03
0.1111110E+01,0.4722E+02
END

```

**FORMAT 2:** FEMB write out format (can be read in also)

```

1  (CURVE NUMBER, I5)
   27 'fld' 100 (num. of points, 'curve name', curve id) (4x,i,'a8',i4)
-4.114211E-01  11.0016E-01
-4.462870E-01  8.52777E-01
-4.004780E-01  8.06437E-01
-3.566750E-01  7.61400E-01
-3.147110E-01  7.17115E-01
 1.110620E-01  5.30703E-01
 2.151110E-01  5.35683E-01
 2.311017E-01  5.31184E-01
 2.623640E-01  5.43322E-01
 2.851711E-01  5.46232E-01
 3.074850E-01  5.48661E-01
 3.211304E-01  5.50701E-01

```

**FORMAT 3:** DYNA format

```

$-----1-----2-----3-----4-----5-----6-----7-----8
$
$           ETA/DYNAFORM USER CURVE
$
$-----1-----2-----3-----4-----5-----6-----7-----8
*DEFINE_CURVE
$CURVETYPE: FLC_CURVE
$  LCID   SIDR   SCLA   SCLO   OFFA   OFFO
    101     0

```

```

$          A1          O1
          -0.3000      0.9918
          -0.2800      0.9363
          -0.2600      0.8829
          -0.2400      0.8317
           0.0600      0.4316
           0.0800      0.4449
           0.1000      0.4575
           0.1200      0.4694
           0.1400      0.4807
           0.1600      0.4912
           0.1800      0.5011
           0.2000      0.5103
           0.2200      0.5188
           0.2400      0.5266
           0.2600      0.5338
           0.2800      0.5402
           0.3000      0.5460
$-----1-----2-----3-----4-----5-----6-----7-----8
*END

```

The dialog box is shown in Figure 11.13.9. eta/DYNAFORM lists all curve data files with the suffix .cur in the FILES field. The user may also read other curve data files by entering the file names.

### 11.13.6 RENUMBER LOAD CURVE

This function enables the user to renumber the ID of a selected curve.

- Select a curve from the SELECT CURVE dialog box shown in Figure 11.13.10.

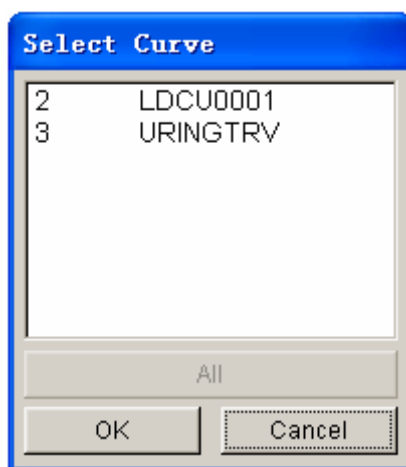


Figure 11.13.10 Select Curve for Renumber

- Enter the new number in the following dialog box.

### 11.13.7 RENAME CURVE

This function enables the user to rename the load curves individually.

- Select a load curve from the list in the SELECT CURVE dialog box.
- Enter the new name in the next dialog box.

### 11.13.8 SHOW LOAD CURVE

This function enables the user to display the selected curve(s). In addition, the user can organize the display by using the options in the dialog box as shown in Figure 11.13.11.

Figure 11.13.11 Show Curve Options dialog box

Select the load curve from the SELECT CURVE dialog box shown in Figure11.13.10.

#### **LOCATION (SHOW CURVE)**

- Toggle on FULL SCREEN to display the curve(s) in the display area.
- Toggle on USER DEFINED LOCATION and drag a window to display the curve(s).

#### **CLEAR**

This function is used to remove an object from the display area using the following options:

- **CURVE SCREEN**  
Clear all objects in the display area.
- **MODEL**  
Clear the displayed model.
- **ETA LABEL**  
Clear the label at the left bottom of the display screen.
- **ALL**  
Clear all objects in the display screen.

*NOTE: Clicking on the CLEAR button removes all objects.*

#### **QUERY**

This function is used to display graph information using the following options:

- **POINT**  
Select a point on a curve. Its X, Y coordinates are printed in the message prompt window.
- **CURVE**  
A list of defined curves appears. Select one from the list or choose from the screen. The name of the curve, number of points on the curve, the X range, and the Y range are printed in the message prompt window.
- **GRAPH**  
The name of the graph and the number of curves in the graph are printed in the message prompt window.

#### **OPTION FOR GRAPH DISPLAY**

This allows the user to manipulate the display of AXIS, GRAPH, CURVE, GRID and

LEGEND. See Figures 11.13.12 ~ 11.13.16.

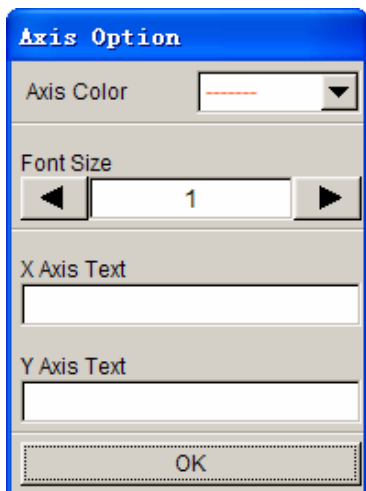


Figure 11.13.12 Axis Options

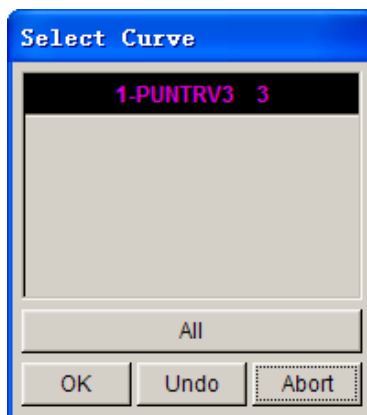


Figure 11.13.13 Select Curve

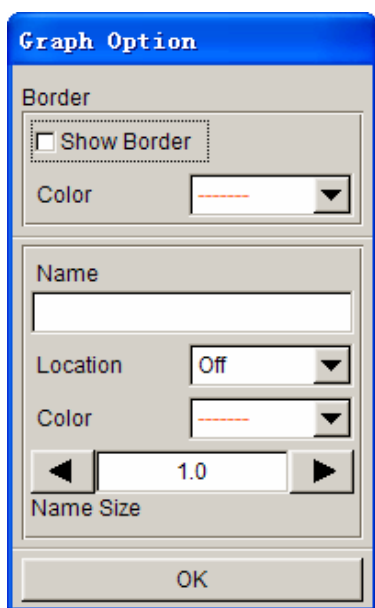


Figure 11.13.14 Graph Options

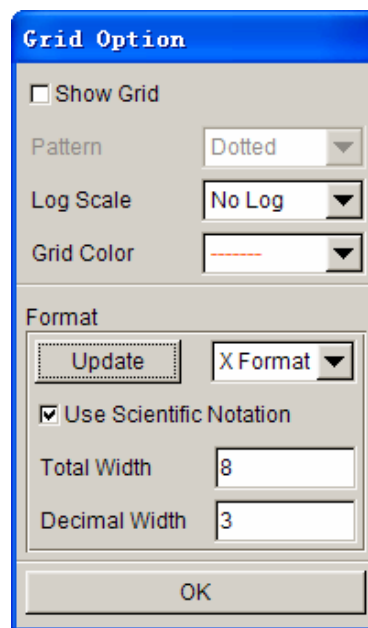


Figure 11.13.15 Grid Options

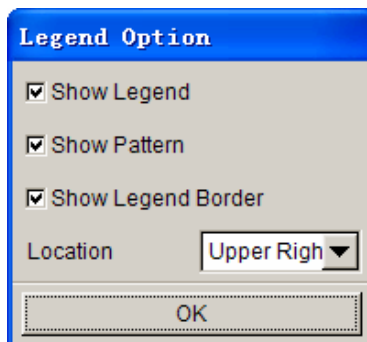


Figure 11.13.16 Legend Options

Select the button to the right of OPTION FOR to select between AXIS, CURVE, GRAPH, GRID and LEGEND. Each option displays its own dialog box.

### DEFINE RANGE OF GRAPH

This function is utilized to zoom in on a specific section of the graph.

The FULL RANGE option is used to display a curve in full range (default).

The USER DEFINED option is used to adjust the values of the four sliders. Select APPLY to reset the range.

### OPERATION

This function allows the user to apply mathematical operations to the displayed curve. The options are shown in Figure 11.13.17.

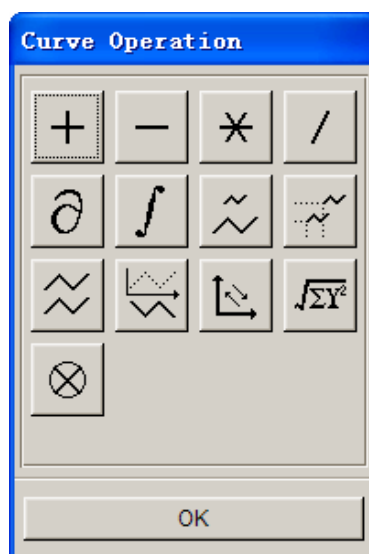


Figure 11.13.17 Curve Operation dialog box

After an operation is selected, the SELECT CURVE dialog box as illustrated in Figure 11.12.18 is displayed. The user can select a curve by choosing from those displayed in the curve window, selecting from the name list, or selecting the ALL CURVES button. eta/DYNAFORM prints the following message in the message prompt window:

***LEFT BUTTON SELECT, MIDDLE BUTTON DESELECT***

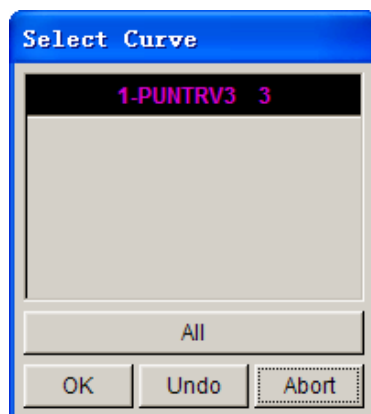


Figure 11.13.18 Select Curve dialog box



Create a new curve in the curve screen of which the values of Y are the sums of the selected curves. At least two curves must be selected.



Create a new curve of which the values are the differences of two selected curves.



Create a new curve of which the values are the product of two curves.



Create a new curve of which the values are quotients of two selected curves.



Create a new curve that is the derivative of the selected curve with respect to time.



Create a new curve that is the indefinite integral of the selected curve with respect to time.



Scale the selected curve using magnification factors in the X, Y directions.



Move the selected curve in the X, Y directions.



Create a new curve that is a copy of the selected curve at the same position and in different colors.



Negate the values of the selected curve in the Y direction.



Switch the X-axis and the Y-axis.



Create a new curve that is the resultant of the selected curve.



Create a new curve of which the abscissa is from the first curve and the ordinate is from the second curve.

**NOTE:** All results under the above operations are temporarily displayed in the curve screen. They will not be automatically saved as load curves. The user needs to save the curve.

### ERASE

This function is used to remove the selected curve from the displayed graph

### KEEP IN DATABASE

This function is used to save the selected curve in the database with a name that begins with CUR.

### SAVE TO FILE

This function is used to save the display graph as a text file in a directory. The SAVE CURVE window is shown in Figure 11.13.19.

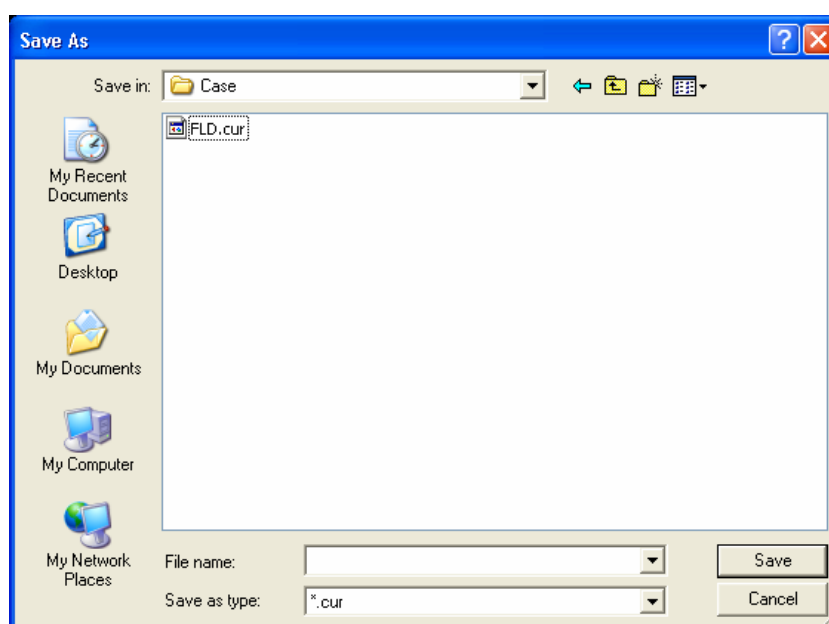




Figure 11.13.19 the Save Curves Window

### 11.13.9 D3PLOT CURVE

This function enables user to create and edit the D3plot curve which is used to control the output frequency of the D3plot file. The D3PLOT CURVE dialog box as illustrated in Figure 11.13.20 is displayed.

**NOTE:** *The D3plot Curve function is available only in Traditional Setup and the Quick Setup (QS).*

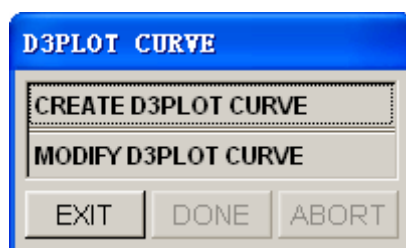


Figure 11.13.20 D3plot Curve dialog box

#### CREATE D3PLOT CURVE

If no D3plot curve is defined in database, the user can use this function to create a new D3plot curve. The D3plot File Number dialog box is displayed after the user selected the CREATE D3PLOT CURVE function. The user can input the Number of the D3plot file in the input field. The default number is 20.

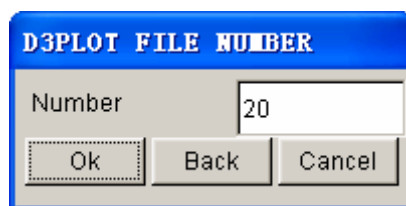


Figure 11.13.21 Define the D3plot file number

The user can click on the OK button to accept the number of D3plot file or clicking on the Back button or Cancel button to exit the D3PLOT FILE NUMBER dialog box.

**NOTE:** *After input the number of D3plot file, the program calculates the time interval and creates a D3plot curve automatically.*

#### MODIFY D3PLOT CURVE

If a D3plot curve is defined in database, the user can edit this curve through the MODIFY D3PLOT CURVE function. If more than one D3plot curves are defined in database, only the last

defined D3plot curve can be edited. The dialog box is show in Figure11.13.22.

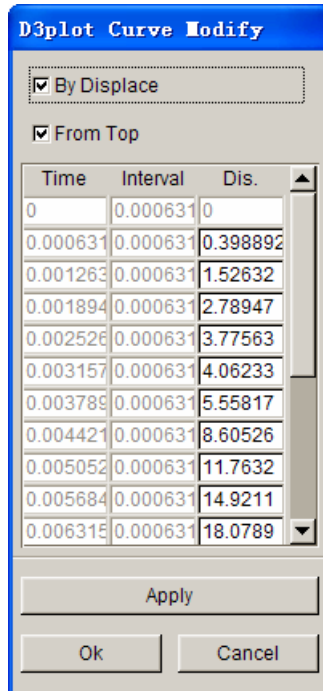


Figure 11.13.22 D3PLOT curve edit dialog box

By default, the D3plot curve is edited through modifying the displacement of the moving tool. Alternatively, the D3plot curve can be edited through modifying the time interval of every D3plot file.

**NOTE:** *The moving tools are related to the setting of Draw Type. If the Draw Type is set as Single Action, the moving tool is the DIE. If the Draw Type is set as Double Action, the moving tool is the PUNCH. The user can change the Draw Type from the menu Tools/Analysis Setup.*

**BY DISPLACE:** This option enables user to edit the D3plot curve through modifying the displacement of the moving tool.

**BY TIME:** This option enables user to edit the D3plot curve through modifying the time of the moving tool.

**FROM TOP:** This option is used usually with the BY DISPLACE option. If this option is toggled on, the displacement means from begin to the current position. If this option is toggle off, the displacement means from the current position to the close position.

**APPLY:** Accept the modification.

**OK:** Exit the D3plot dialog box and accept the modification.

**CANCEL:** Exit the D3plot dialog box without saving.

## 11.14 SHOW LINE

This function is used to identify any existing line and its direction. Refer to Section 5.1 for detailed information.

## 11.15 COORDINATE SYSTEM

The functions in this menu are used to create and modify local coordinate systems. The functions are shown in Figure 11.15.1.

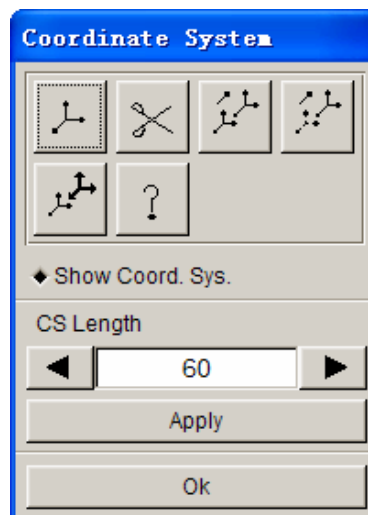


Figure 11.15.1 Coordinate System dialog box

Click on the arrow buttons to modify the size of the displayed coordinate system. Click on the APPLY button to refresh the display of coordinate system.

- **Show Coord. Sys.**

Display all created coordinate systems.

A detailed description of these functions is given in the following sections.

### 11.15.1 CREATE COORDINATE SYSTEM



This function is used to create a local coordinate system.

eta/DYNAFORM refers to the local coordinate system to translate, rotate, mirror, copy, and generate points, lines, or nodes. When the function is selected, the user needs to generate a local system designated as the UVW coordinate frame using the LCS dialog box as illustrated in Figure 11.15.2. The user can create a new LCS or select a coordinate system as the current LCS by selecting GLOBAL, CURRENT LCS, LAST, or VIEW DIRECTION.

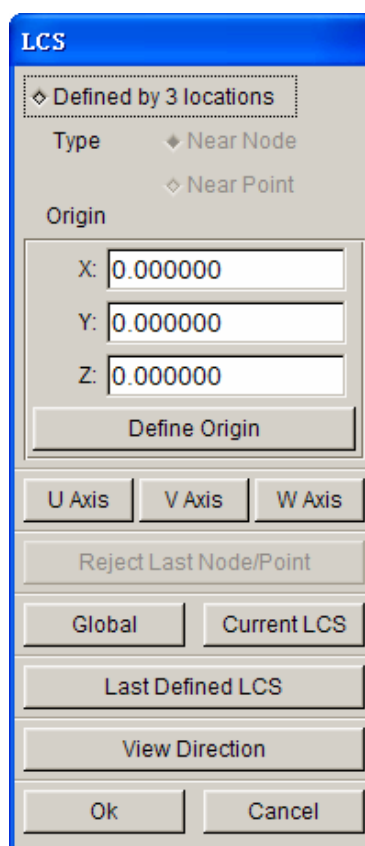


Figure 11.15.2 LCS dialog box

### DEFINED BY 3 LOCATIONS

This function enables user to define a local coordinate system through specifying three nodes/points.

**Near Node:** Define LCS by selecting three nodes. The first selected node will be defined the origin of the LCS. The coordinate value of the defined origin in Global system is also shown in below. User can edit it manually.

**Near Point:** Define WCS by selecting three geometry points. The first selected point will be defined the original of the LCS.

**Reject Last Node/Point:** Reject last selected node/point.

*NOTE: If the user defines a LCS through the DEFINED BY 3 LOCATIONS option, at least one node/point need defined.*

*If only one node/point defined, the origin of the LCS is the defined node/point. The local U, V, W axis are parallel to the global X, Y, Z axis respectively.*

*If only two nodes/points defined, the origin of the LCS is the first selected node/point. The local W axis is the vector orientation from the first node/point to the second node/point.*

*If three nodes/points defined, the origin of the LCS is the first selected node/point. The local U axis is the vector orientation from the first node/point to the second node/point. The selected three nodes/points decided the local U-V plane. The local W axis is vertical with the U-V plane.*

### THROUGH U, V, W

This function enables user to define a local coordinate system through specifying the local U, V and W axis.

**U Axis...:** This function allows the user to define local U axis direction. After clicking this button, the DEFINE DIRECTION dialog box is displayed. The user may select any of the LCS in the list.

**V Axis...:** This function allows the user to define local V axis direction. After clicking this button, the DEFINE DIRECTION dialog box. The operation is the same as in U axis.

**W Axis...:** This function allows user to define local W axis direction. After clicking this button, the DEFINE DIRECTION dialog box is displayed. The operation is the same as U and V axes.

*Note: The user may define U axis and V axis to define LCS. W direction is calculated according to the right hand rule.*

$$W=U \times V$$

*The V axis is adjusted again from the W and U axis according to the right hand rule.*

$$V=W \times U$$

*The user may only define W axis direction to define the LSC. The U, V direction is defined automatically. This option is useful when the local U and V direction are not important.*

### GLOBAL

Create a local coordinate system using the global coordinate system and set the new created LCS as the current LCS.

**CURRENT LCS**

Create a local coordinate system using the current LCS.

**LAST DEFINED LCS**

Create a local coordinate system using the last defined coordinate system and set the new created LCS as the current LCS..

**VIEW DIRECTION**

Define the view directions as the W-Axis of the current LCS. The user can select or define a point as the origin, or the origin of the global coordinate system can be the origin of the current LCS.

The DEFINED BY 3 LOCATIONS option enables the user to create a local coordinate system based on the selection of three reference points/nodes.

**11.15.2 DELETE COORDINATE SYSTEM**

This function enables the user to remove selected local coordinate system from the database. The SELECT LCS dialog box as illustrated in Figure 11.14.3 is displayed.

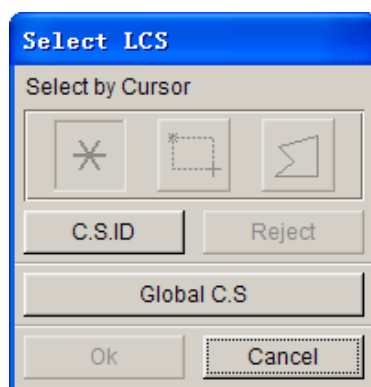


Figure 11.15.3 Select LCS



Pick a coordinate system.



Drag a window to include the coordinate system.



Define a multi-point region to include the coordinate system.

**C.S. ID**

Enter the starting and ending coordinate system ID and the increment in the following dialog box.

**REJECT**

Reject the last selection.

**NOTE:** *To remove the local systems/definitions from individual nodes, the user must reassign the nodes to the GLOBAL COORDINATE SYSTEM.*

**11.15.3 COPY COORDINATE SYSTEM**

This function enables the user to copy and create a selected coordinate system in a new location.

- The SELECT LCS dialog box provides three options: 1) Pick the coordinate system displayed on the screen using the default mouse picks, 2) Select C.S ID to select an LCS by the given ID, 3) Select GLOBAL C.S. to select the GLOBAL COORDINATE SYSTEM.
- Define the new location using the functions provided in the INPUT COORDINATE dialog box. The selected coordinate system is copied automatically.

**11.15.4 MODIFY COORDINATE SYSTEM**

This function is used to modify the definition of an existing coordinate system.

After a coordinate system is selected from the SELECT LCS dialog box, the SELECT OPTION dialog box as shown in Figure 11.15.4 is displayed.

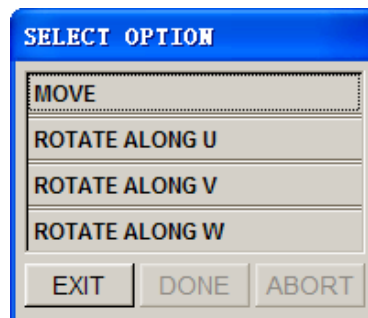


Figure 11.15.4 Modify C.S Options

If the MOVE function is selected, the INPUT COORDINATE dialog box is displayed. The user needs to define the new location of the coordinate system. The coordinate system is moved according to the defined location.

The user can rotate the selected coordinate system about U, V, or W axis. If the ROTATE function is selected, the ROTATE ANGLE dialog box is displayed. The user needs to key in the angle of rotation. The coordinate system is rotated according to the input angle of rotation.

### 11.15.5 CURRENT COORDINATE SYSTEM



This function allows the user to assign a CURRENT status to a selected coordinate system.

Select a coordinate system using mouse cursor; the selected coordinate system is highlighted and the color of previous coordinate system is changed.

### 11.15.6 IDENTIFY COORDINATE SYSTEM



This function is used to identify the type and origin of a coordinate system.

Select a coordinate system using mouse cursor; eta/DYNAFORM prints the following message in the message prompt window:

**C.S. #### [SYSTEM TYPE] ORIGIN (x y z)**

Where:

#### is the coordinate system numbers.

SYSTEM TYPE is the system type (rect., sphere, cylin.), and

ORIGIN is the global location of the system origin.

## 11.16 DATABASE STATISTIC

This function enables the user to view the statistics of the eta/DYNAFORM database regarding geometry, models, materials and interfaces. The DATABASE STATISTICS window is illustrated in Figure 11.16.1.



Database Statistics	
— Geometry Information —	
Lines	0
Points	0
Surfaces	0
Edge Points	0
Surface Boundaries	0
Control Points	0
Real Parameters	0
Integer Parameters	0
— Model Information —	
Nodes	8779
Max Node ID	10987
Elements	8348
Max Element ID	10532
Materials	1
Properties	1
Parts	10
Max Part ID	14
Local Coordinates	5
— Element Information —	
Shell	8348
Solids	0
Beams&Rods	0
Nodal Rigid Bodies	0
Spot Welds	0
Spring/Dampers	0
Masses	0
Thick Shells	0
— Tool Information —	
Blank	Defined
Punch	Defined
Die	Defined
Binder	Defined
User Defined Tools	4
— Other Information —	
Nodes with Initial Velocity	0
Drawbeads	0
— Shell Element Information —	
Quad.Elements	8236
Tri.Elements	112
— Solid Element Information —	
Hexahedron Elements	0
Wedge Elements	0
Tetrahedron Elements	0
Ok	

Figure 11.16.1 Database Statistics window

## Chapter 12

# VIEW OPTIONS

The functions provided in this menu are utilized to adjust the graphic display of the items shown in the display window of eta/DYNAFORM. The options are shown in Figure 12.1.

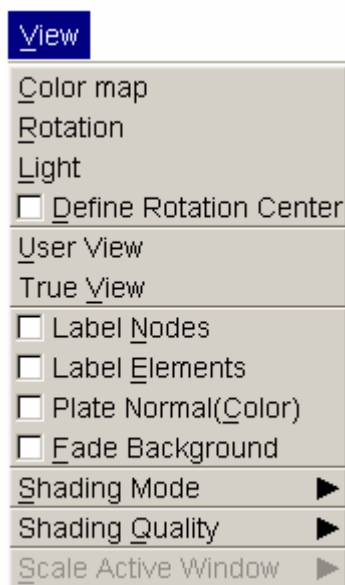


Figure 12.1 the View Options Menu

A detailed description of each option is given in the following sections.

### 12.1 COLOR MAP

This function is used to change the default color in the color bar. The dialog box is illustrated in Figure 12.1.1.

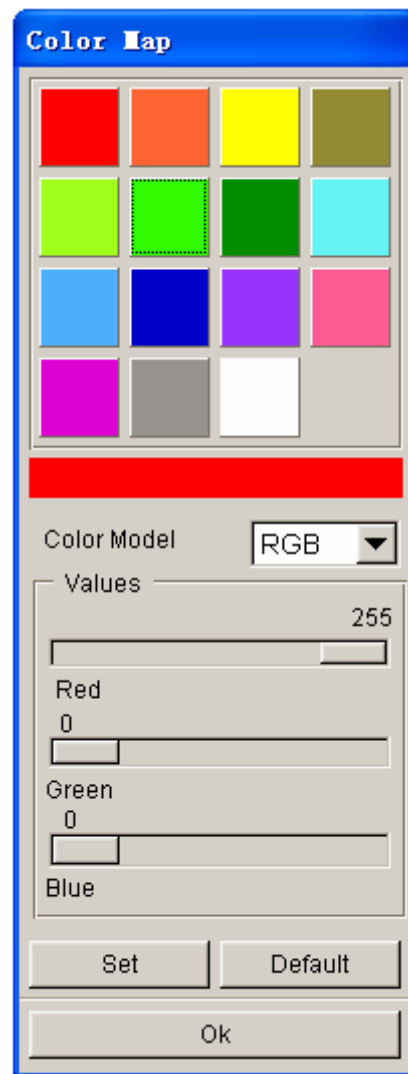


Figure 12.1.1 the Color Map dialog box

- 15 colors are available from the color palette. The selected color is shown in the long rectangular bar.
- The COLOR MODEL function provides two options: RGB (Red, Green, and Blue) and HSV (Hue, Saturation, and Luminosity).
- The sliders enable the user to change the values of RGB and HSV.
- SET confirms the change of values for the color model.
- DEFAULT resets the values of the color model.

## 12.2 ROTATION

This function enables the user to rotate an object in the display area about the global (or virtual) X, Y, and Z-axes. The options are shown in Figure 12.2.1.

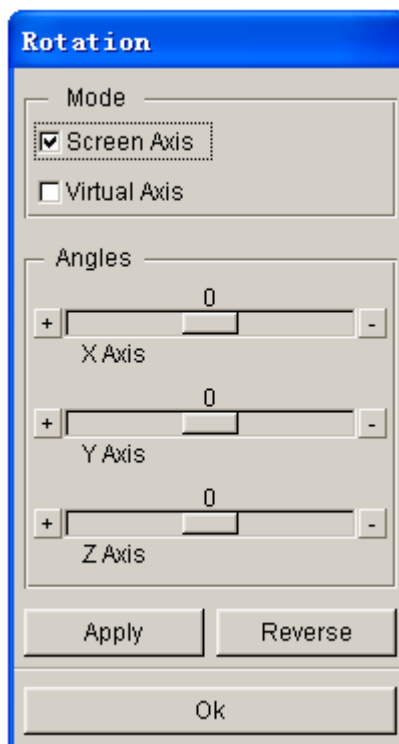


Figure 12.2.1 Rotation dialog box

- **SCREEN AXIS**

Rotate an object in the display area according to a specific rotation angle. The screen axes are fixed at all times. They are defined as the following:

1. **SCREEN X**

Axis direction is from the left to right of the terminal screen.

2. **SCREEN Y**

Axis direction is from the bottom to the top of the terminal screen.

3. **SCREEN Z**

Axis direction is from the screen to the user.

- **VIRTUAL AXIS**  
Rotate an object about global (or virtual) X, Y, and Z-axes according to an angle.
- The sliders enable the user to adjust rotation angles.
- Click on the APPLY button to complete the rotation. This function can be repeated as many times as desired.
- Click on the REVERSE button to reverse the rotation. This function can be repeated as many times as desired.

## 12.3 LIGHT

This function enables the user to uniformly move a light source direction with a consistent intensity. The dialog box is shown in Figure 12.3.1. The right and left arrow buttons are utilized to move the light source in the positive and negative X-direction, respectively. The up and down arrow buttons are utilized to move the light source in the positive and negative Y-direction, respectively.

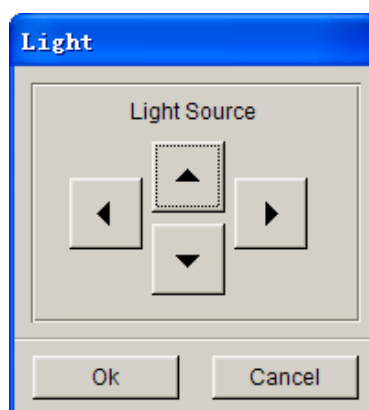



Figure 12.3.1 The Light Source dialog box.

## 12.4 DEFINE ROTATION CENTER

If this option is toggled on, the user can define the rotation center prior to rotating the model. When the user clicks the  button, eta/DYNAFORM prints the following message in the message prompt window:

***PICK A LOCATION AS THE ROTATION CENTER***

The user can click anywhere in display window as the rotation center. Then, move the mouse cursor to

rotate the model around the center.

If the user uses the shortcut key (CTRL+ left mouse button) to rotate the model, the immediate location upon which the mouse cursor is placed is automatically defined as the rotate center. The model is rotated around the center.

## 12.5 USER VIEW

This function is utilized to save/store a desired view in the current eta/DYNAFORM database. The saved/stored views are recalled. The USER VIEWS dialog box is illustrated in Figure 12.5.1.

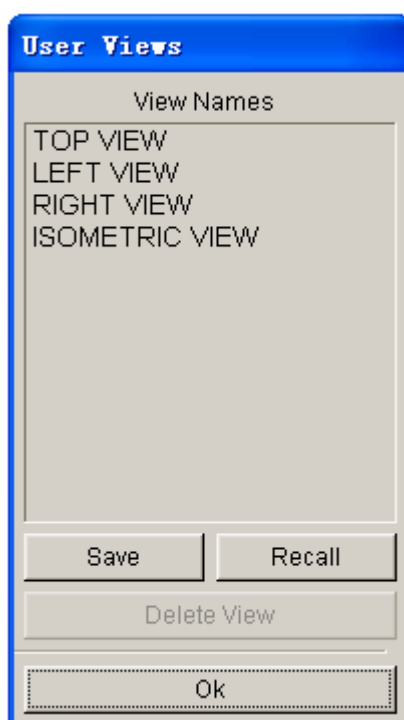


Figure 12.5.1 User Views dialog box

- **SAVE**

Save the current displayed view in the eta/DYNAFORM database. The saved view is listed in the window. eta/DYNAFORM allows the user to save up to six user defined views plus the four pre-defined views (a total of 10 views). The user can type in the name of each saved view using the VIEW TITLE dialog box as illustrated in Figure 12.5.2.

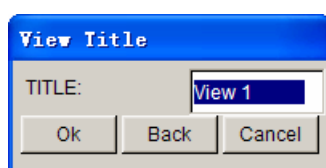


Figure 12.5.2 Save View dialog box

- **RECALL**  
Display the selected view from the list.
- **DELETE VIEW**  
Remove the selected view from the list.

## 12.6 TRUE VIEW

This function enables the user to display an object in true view, i.e. the normal view of the local W-axis as projected onto the local UV plane.

- The LCS dialog box is displayed to enable the user to define a local coordinate system.
- The eta/DYNAFORM QUESTION dialog box as shown in Figure 12.6.1 is displayed.

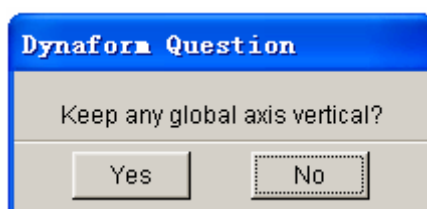


Figure 12.6.1 DYNAFORM Question dialog box

1. YES  
Allow the user to select an axis to be displayed from the bottom to the top of screen. The object remains in its true view if this option is selected.
2. NO  
Display the object in its true view (perpendicular to the W-axis).

## 12.7 LABEL NODES (toggle)

This function enables the user to show/hide the node labels. The node number of all displayed nodes is shown when the function is toggled on. This function is also found in PREPROCESS/NODE menu.

## 12.8 LABEL ELEMENTS (toggle)

This function enables the user to show/hide the element labels. The element number of all displayed elements is shown when the function is toggled on. This function is also found in PREPROCESS/ELEMENT menu.

## 12.9 PLATE NORMAL (Color)

This function enables the user to display the normal direction of the elements by color.

## 12.10 FADE BACKGROUND

This function enables user to change the background color of display window in eta/DYNAFORM. The default background color is black. If this option is toggled on, the background is changed to fade blue as illustrated in Figure 12.10.1.

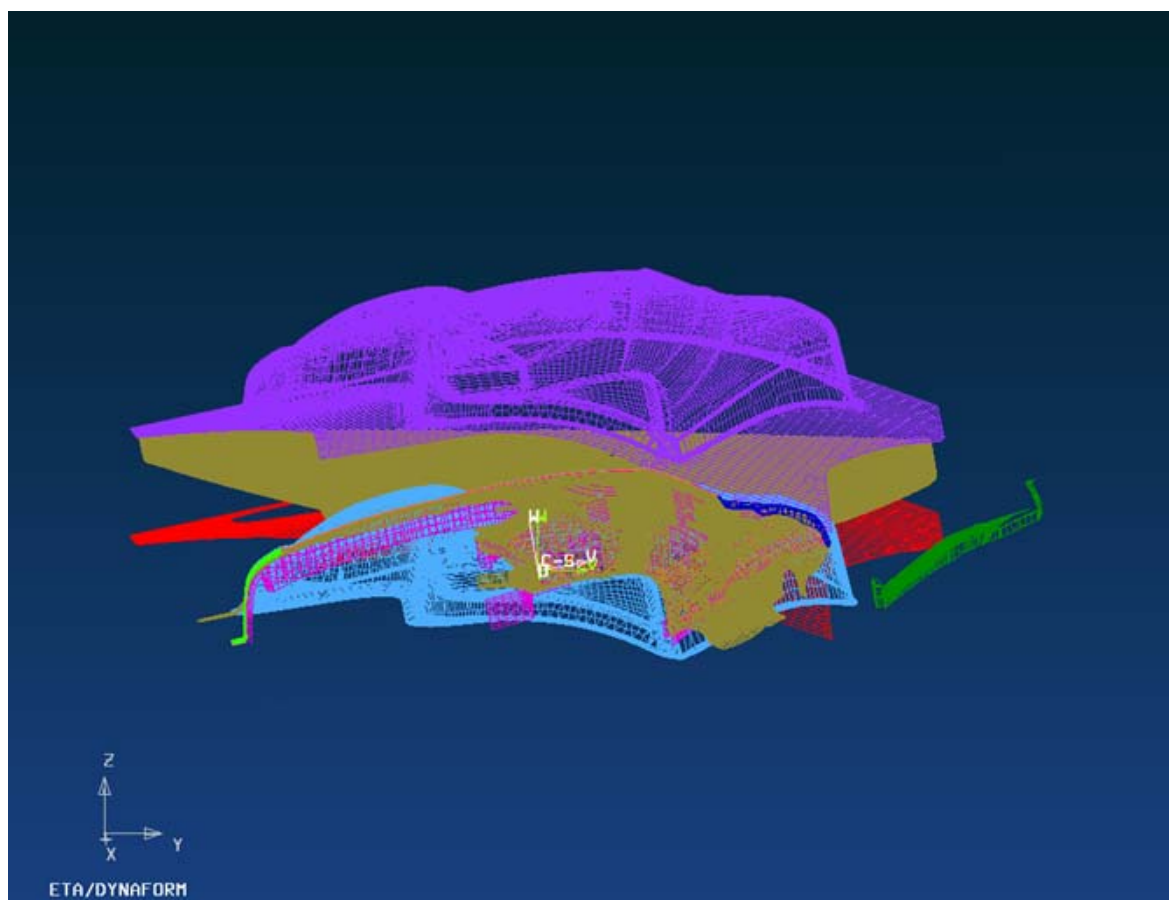


Figure 12.10.1 Fade background



## 12.11 SHADING MODE

As shown in Figure 12.11.1, the SHADING MODE offers three options for shading a part or a model. The GOURAND SHADDING mode (smooth shading) is the default setting for eta/DYNAFORM and is based on average nodes algorithm. The FLAT SHADING mode is based on elements.

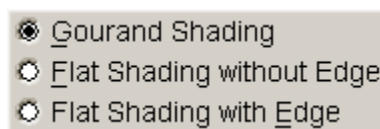


Figure 12.11.1 Shading Mode Menu

## 12.12 SHADING QUALITY

As shown in Figure 12.12.1, SHADING QUALITY offers three options for shading a part or a model. The HIGH mode has the best shading quality but requires more CPU time. The NORMAL mode is the default setting for shading. The LOW mode has an inferior shading quality but is the quickest mode when shading. The user can adjust this parameter according the performance of the computer or the need for high resolution images.

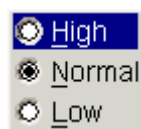


Figure 12.12.1 Shading Quality Submenu

## 12.13 SCALE ACTIVE WINDOW

This function enables the user to scale an active window display by the scale factor provided from the pull-down menu. When this function is active, only the active window function is performed.

## Chapter 13

### ANALYSIS

The functions provided in the ANALYSIS menu, as illustrated in Figure 13.1 allow the user to submit an analysis or to generate an output file.



Figure 13.1 Analysis menu

A detailed description of each function in the menu is given in the following sections.

### 13.1 LS-DYNA (CTRL+A)

If the draw type is not set as SPRINGBACK or GRAVITY ONLY (Tools/Analysis Setup/Draw type), the ANALYSIS dialog box is shown in Figure 13.1.1. The LS-DYNA menu allows the user to set the parameters for stamping simulation, gravity loading and springback analysis. These parameters are written into the analysis output files. The default dialog box of this menu (for stamping simulation) is shown in Figure 13.1.1.

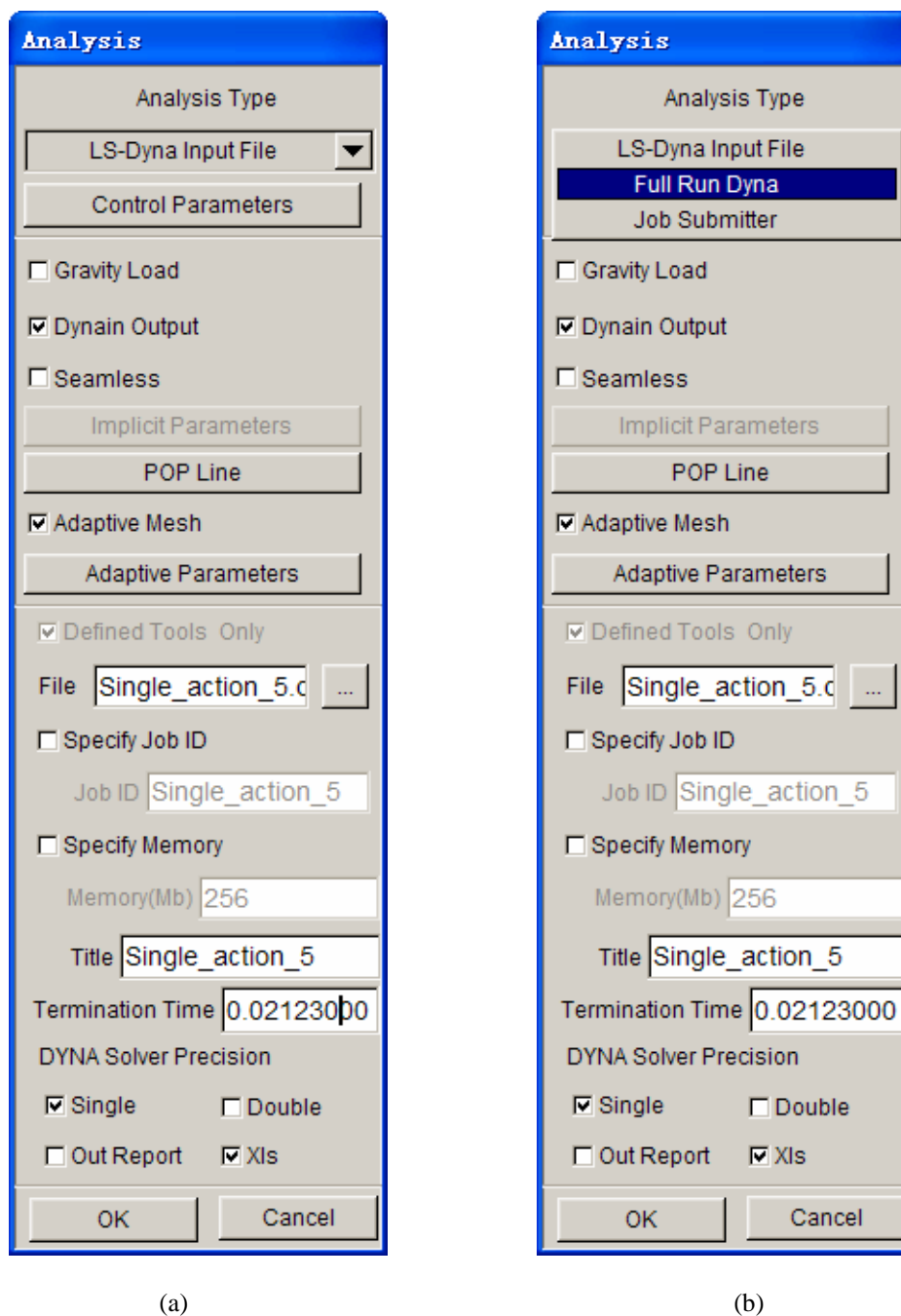


Figure 13.1.1 Analysis dialog box (a) LS-DYNA Input (b) Full Run DYNA

### 13.1.1 ANALYSIS TYPE

There are three types of analysis output:

1. LS-DYNA INPUT FILE

This function outputs the setup results as an LS-DYNA input deck.

2. FULL RUN DYNA

This function outputs a LS-DYNA input deck and launches the LS-DYNA to run the current job immediately.

**NOTE:** The user should set the path of the LS-DYNA solver in Tools/Analysis Setup menu.

3. JOB SUBMITTER

This function outputs a LS-DYNA input deck and submits it to Job Submitter. Then, the user can manage the job in Job Submitter. The GUI is illustrated in Figure 13.1.2.

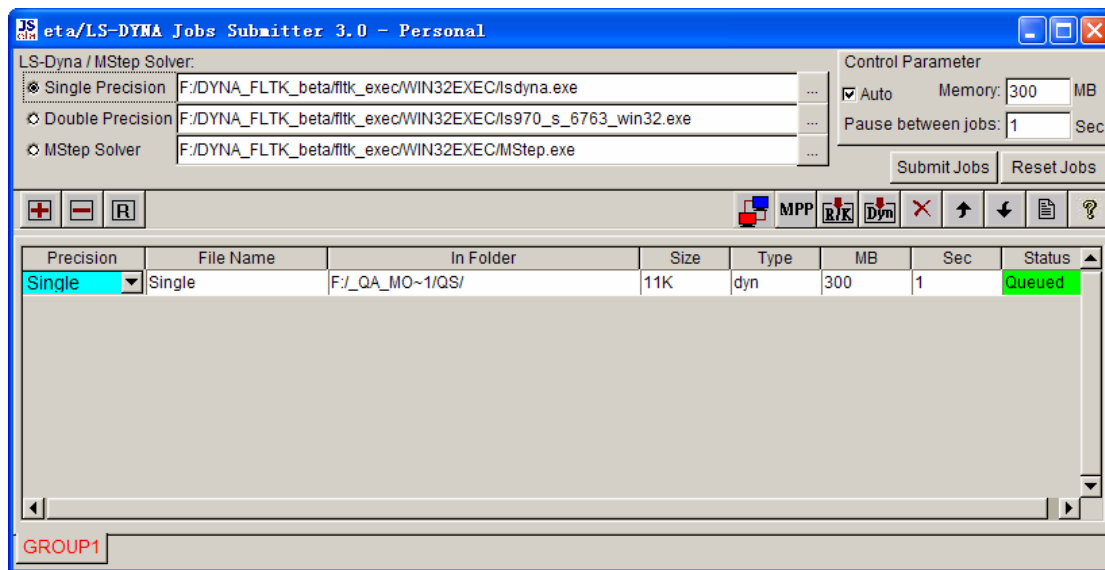


Figure 13.1.2 Job Submitter GUI

The user can select the type of analysis output by clicking on the toggle menu.

### CONTROL PARAMETERS

This function defines the control parameter values for running a stamping simulation using LS-DYNA. The user can edit the parameters by clicking on the CONTROL PARAMETERS icon from the ANALYSIS dialog box. The DYNA3D CONTROL PARAMETER dialog box is shown in Figure 13.1.3.

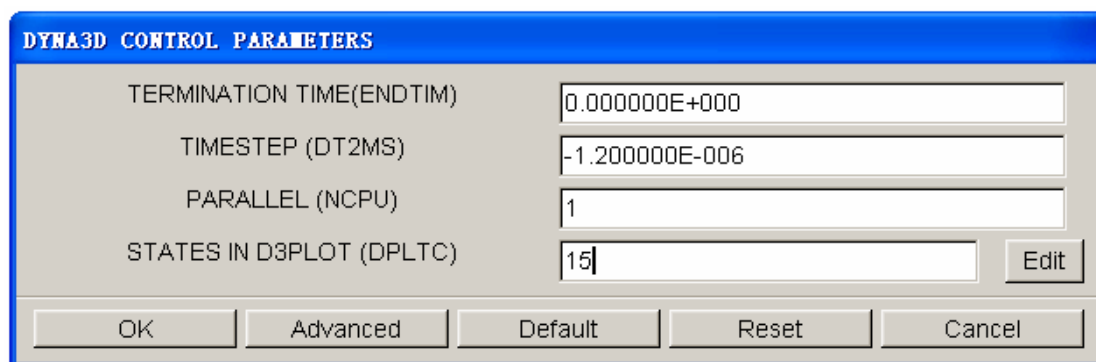


Figure 13.1.3 Control Parameters dialog box

**TERMINATION TIME**

Determine by the time duration of motion curve.

**TIMESTEP (DT2MS)**

Time step size for mass scaling. Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_TIMESTEP for more information.

**PARALLEL (NCPU)**

This setting allows the user to define the number of processors to be used by the solver. This function requires PC hardware with multiple processors, a shared memory (parallel process) version of the LS-DYNA solver, and valid license. Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_PARELLEL for more information.

**STATES IN D3PLOT (DPLTC)**

This setting allows the user to define the number of evenly spaced frames of data, displayed and animated in eta/POST. Please refer to the *LS-DYNA Keyword User's Manual*, \*DATABASE\_BINARY\_D3PLOT for more information.

**ADVANCED OPTIONS****HOURLASS (IHQ)**

Optional hourglass force vector modifier. Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_HOURLASS for more information.

**CONTACT (SLSFAC)**

Scale factor for sliding interface penalties. Please refer to the *LS-DYNA Keyword User's*

*Manual*, \*CONTROL\_CONTACT for more information.

#### **CONTACT (ISLCHK)**

Initial penetration check in the contact surfaces. Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_CONTACT for more information.

#### **CONTACT (SHLTHK)**

Shell thickness considered in SURFACE-TO-SURFACE and NODE-TO-SURFACE contact options. Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_CONTACT for more information.

#### **CONTACT (PENOPT)**

Penalty stiffness value option. Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_CONTACT for more information.

#### **TERMINATION (ENDMAS)**

Percent change in the total mass for termination of calculation. Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_TERMINATION for more information.

#### **INTEGRATION (MAXINT)**

Number of shell integration points. Please refer to the *LS-DYNA Keyword User's Manual*, \*DATABASE\_EXTENT\_BINARY for more information.

#### **OUTPUT TIMES (ENDTIM/DT)**

Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_TERMINATION for more information.

#### **ACCURACY STRESS (OSU)**

Global flag for objective stress updates. Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_ACCURACY for more information.

#### **ACCURACY NODE (INN)**

Invariant node numbering for shell elements. Please refer to the *LS-DYNA Keyword User's Manual*, \*CONTROL\_ACCURACY for more information.

**OK** - Accepts the control parameters.

**DEFAULT** - Resets all values to their default settings.

**RESET** - Resets the last defined setting to its previously defined value.

**ADVANCED** - Extends the table to make the advanced control parameters show up.

**CANCEL** - Aborts the table and resets all values to their previous settings.

*NOTE: Refer to the LS-DYNA User's Manual for detailed information about each control parameter.*

### 13.1.2 GRAVITY LOAD (toggle)

This function toggles gravity force ON/OFF and writes out the relative control cards to the input deck.

### 13.1.3 DYNA OUTPUT (DYNAIN) (toggle)

This function automatically writes out the INTERFACE\_SPRINGBACK\_DYNA3D card to the input deck. Therefore, LS-DYNA will create a DYNAIN file at the end of the simulation run. The DYNAIN file contains the deformed blank results, including thickness, stress and strain.

### 13.1.4 SPRINGBACK (SEAMLESS)

This function toggles on/off the LS-DYNA seamless explicit/implicit switch. It will activate implicit springback analysis after the stamping simulation. If SEAMLESS is toggled on, the user can set the springback constraint and define the IMPLICIT parameters as shown in Figure 13.1.4.

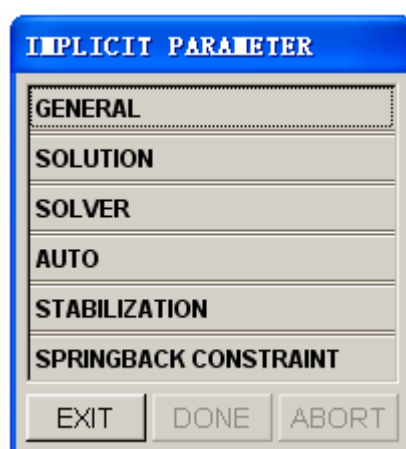


Figure 13.1.4 Implicit Parameters dialog box

**NOTE:** Refer to the LS-DYNA User's Manual, keyword *\*CONTROL\_IMPLICIT\_OPTIONAL*, for more information.

### 13.1.5 POP LINE

This function enables user to define the POP line for the eta/POST. If defined, the line information is output to the index file.

**NOTE:** This function is especially for eta/Post. When eta/Post opens the result file (D3plot), it will also import the related idx file. Then the POP line information will be read by eta/Post.

The user can click on the POP line button to display the POP line dialog box as illustrated in Figure 13.1.5.

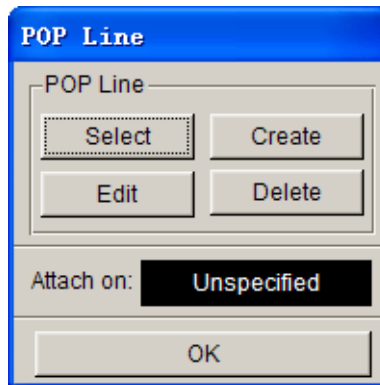


Figure 13.1.5 Define POP LINE dialog box

### SELECT POP LINE

This function enables user to define the POP line through selecting the existing one or multiple lines in current database. The SELECT LINE dialog box is displayed after the user clicked this button. Each time the user enters the POP line dialog box, the defined POP line is highlighted on screen.

### CREATE POP LINE

This function enables user to create the POP line from the Addendum defined in eta/DYNAFORM. Prior to using this function, the user has to create the addendum in eta/DYNAFORM. This function is similar with the function CREATE PO LINE described in Section 6.3.5.



## EDIT POP LINE

This function enables user to edit the defined POP line. After selecting this button, the Curve Edit dialog box is displayed. The user can edit the highlighted POP line. Please refer to the Section 6.2.1 for more details.

## DELETE POP LINE

This function enables user to remove the selected POP line. After selecting this button, eta/DYNAFORM prompts the user to select the highlighted POP line.

*NOTE: The original referenced lines will not be deleted from the database.*

## ATTACH ON

This function enables user to select one Part from the Part list. The defined POP line will be attached on the selected Part. In eta/POST, the POP line moves together with the defined Part. We select the DIE as the Attach part.

After all the parameters of the POP line are defined, the user can click the OK button to exit the POP line dialog box. When a LS-DYNA input deck is output from eta/DYNAFORM, the POP line information is written into the idx file.

### 13.1.6 ADAPTIVE MESH (toggle)

If this function is toggled on, ADAPTIVE scheme is activated during the stamping simulation. The user can edit the adaptive parameters by clicking on the ADAPTIVE PARAMETERS button. The default ADAPTIVE CONTROL PARAMETERS dialog box is shown in Figure 13.1.6. The ADVANCE button enables the user to expand the dialog box for setting other adaptive parameters.

*NOTE: Refer to the LS-DYNA User's Manual, keyword \*CONTROL\_ADAPTIVE, for detailed information about each parameter.*

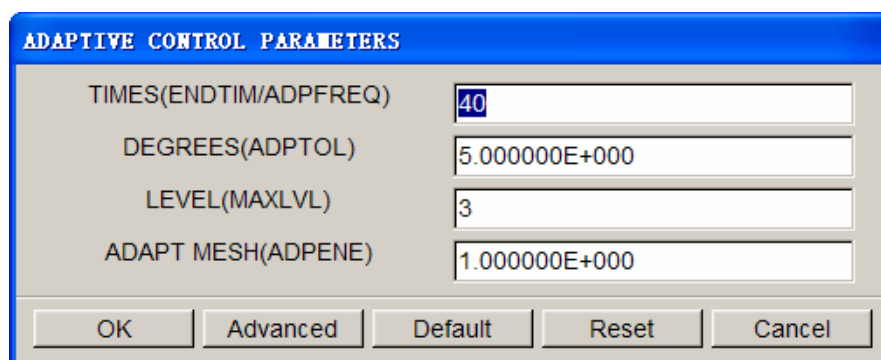


Figure 13.1.6 Adaptive Control Parameters dialog box

### 13.1.7 DEFINED TOOLS ONLY (toggle)

If the option is toggled on, only defined tools are output to the input deck. If the option is toggled off, all parts in the eta/DYNAFORM database are output.

### 13.1.8 SPECIFY OUTPUT FILENAME

The user can specify a file name for the output keyword file. The default keyword file name is same with the name of current database. The user can input the new file name in the FILE NAME FIELD directly or click the button to select the path to save the file.

### 13.1.9 SPECIFY JOB ID (toggle)

This is a new feature for LS-DYNA970 to assign file name prefix to the result files. If the option is toggled on, the input field next to **Job ID** is activated allowing the user to enter the JOB ID for the analysis. Default JOB ID is the database file name without the “.df” extension.

### 13.1.10 SPECIFY MEMORY (toggle)

If this option is toggled on, the user can allocate memory in megabytes (Mb) for running LS-DYNA. This function will automatically convert the input memory in megabytes to memory in words and write it out to the \*KEYWORD card in the input deck.

### 13.1.11 SPECIFY TITLE

Every keyword file should be given a title, which helps the user to understand the content of the analysis when processing the forming results in eta/POST. The default title is the name of the current database.

### 13.1.12 TERMINATION TIME

This option is utilized to display the current termination time. The termination time is calculated by the program, which is based on the tool motion curve defined by the user. Usually, the user needs not to modify time.

### 13.1.13 DYNA SOLVER PRECISION

#### SINGLE

This option is valid only when the analysis type is set as FULL RUN DYNA. If this option is toggled on, the simulation is conducted using the specified single precision solver.

**DOUBLE**

This option is valid only when the analysis type is set as FULL RUN DYNA. If this option is toggled on, the simulation is conducted using the specified double precision solver.

**OUT REPORT**

This option allows the user to output the current basic information, the default file name is the name of the current database. This helps the user to understand the current information, and it is convenient to communication between users. If the XLS option is not selected, the default postfix is CSV, otherwise the postfix is XLS.

**XLS**

This option allows the user output the information file with the postfix: XLS. Figure 13.1.7 shows a typical XLS file.

+ <DW Process>				
User Name	jli			
Date	2006.4.29			
Solver	LS-DYNA(Single Precision)			
Pre-Post	eta/DYNAFORM 5.5			
<b>&lt;Data&gt;</b>				
Case Drive	F:			
Case directory	F:/QA/MD*1/DRAW/B*1/新建文*1			
CAD Data Name				
tools of reference	upper tool			
<b>&lt;Material&gt;</b>				
DB name	BLANKMAT	TYPE	36	
Thickness (mm)	2.00	Die Gap (%)	10	
integration points	5	intergation method	Reduce	
<b>&lt;FE Mesh/Blank&gt;</b>				
Blank Mesh pitch(mm)	Max	3.00	Min	1.00
Adaptive Level(MAXLVL)		3		
<b>&lt;FE Mesh/Tool&gt;</b>				
Max Size(mm)	30.0	Min Size(mm)	0.5	
Chordal Dev.(mm)	0.15	Angle(degree)	20.0	
Gap Tol(mm)	2.5	Ignore Hole Size(mm)	0.0	
Offset(mm)	2.0			
<b>&lt;Draw Type&gt;</b>				
Type	Double action			
Binder control	constant force			
Draw direction	Z			
Cusion Stroke/Lower binder travel(mm)	4.0	tool stroke		
Dies-Binder clearance(mm)	4.00	Dies-Punch clearance(mm)	44.00	
Binder Begin Time		binder close	after binder close	
Cusion Force/Lower Binder Force(N)	0	( 100 ton)		
home clearance(mm)	0.00			
Binder Close Velocity (mm/s)	2000			
Tool Travel Velocity (mm/s)	5000			
<b>&lt;MISC&gt;</b>				
Contact Type	CONTACT_ONE_WAY_SURFACE_TO_SURFACE			
SLSFAC	0.100	DT2MS	-1.200E-06	
Static Friction	0.125	Dynamic Friction	0.00	
Damping Coeff.	20.0			
Memory(MBytes)				

Figure 13.1.7 Typical XLS file

## 13.2 LS-DYNA (SPRINGBACK )

If the draw type is set as SPRINGBACK (TOOLS/ANALYSIS SETUP/DRAW TYPE), the following dialog box is displayed (Figure 13.2.1):

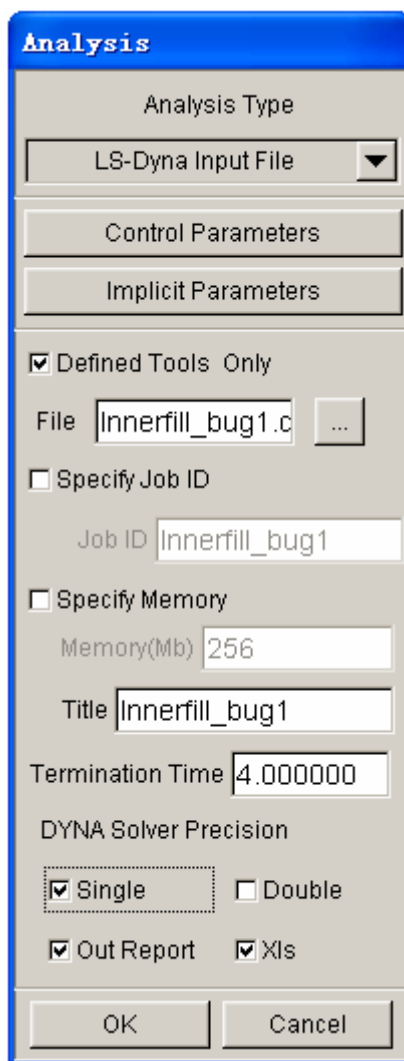


Figure 13.2.1 Springback Analysis dialog box

These functions are discussed in Section 13.1, LS-DYNA.

**NOTE:** Refer to the LS-DYNA User's Manual for more information.

### 13.3 LS-DYNA (GRAVITY LOADING)

If the draw type is set to GRAVITY (TOOLS/ANALYSIS SETUP/DRAW TYPE), the following dialog box is displayed (Figure 13.3.1):

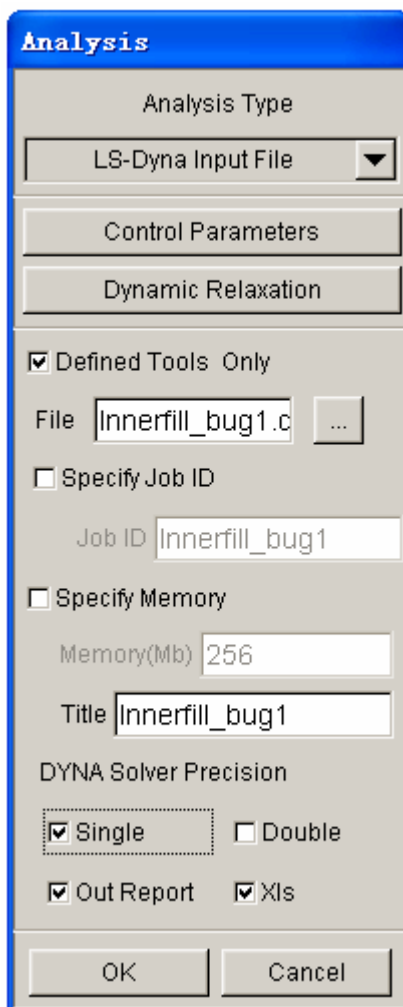


Figure 13.3.1 Gravity Loading dialog box

**NOTE:** Refer to the *LS-DYNA User's Manual*, keyword *\*CONTROL\_DYNAMIC\_RELAXATION*, for detailed information about each parameter.

## 13.4 MSTEP

MSTEP is a brand new, modified-one-step solver adopted by eta/DYNAFORM to quickly estimate formability and the blank outline in the early stages of the stamping design cycle.

Please refer to Section 7.2 for detailed information about MSTEP.

## 13.5 OUTPUT NEW DYNAIN FILE

This function allows the user to output a new DYNAIN file (for example, after trimming).

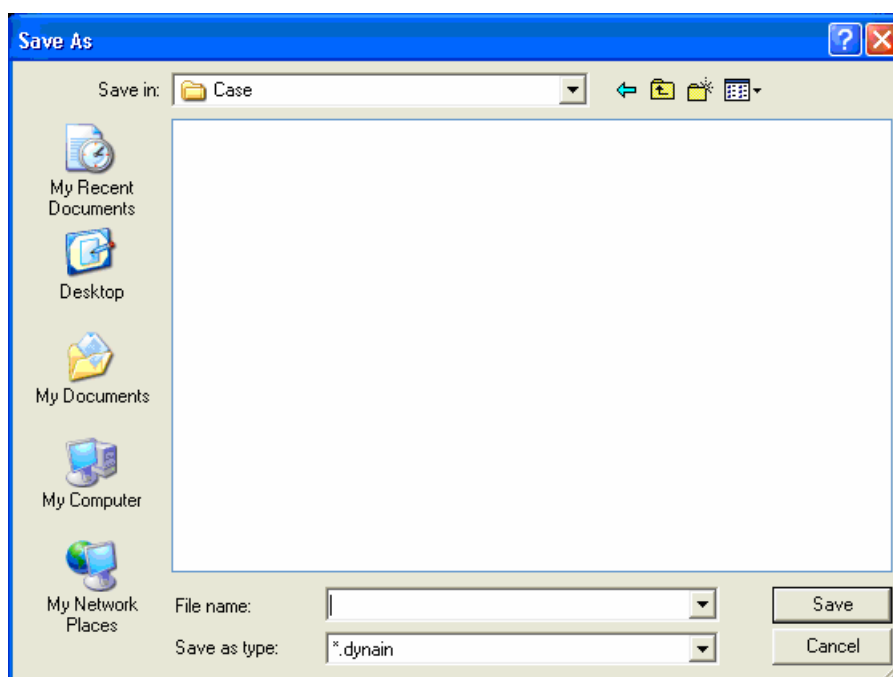


Figure 13.5.1 Output DYNAIN File

APPENDIX A  
**HARDWARE AND SOFTWARE REQUIREMENTS**

**UNIX**

PLATFORM	OS VERSION	GRAPHICS CARD	DISK SPACE (MB)
HP	HP-UX 11+	Minimum CRX 8 Plane	200
IBM	AIX 4.2+	Minimum 24 Plane Graphics	200
SGI	IRIX 6.5+	All Graphics Boards Supported	200
SUN	SunOS 5.8+	Minimum Creator 3D	200

**LINUX**

The RedHat operating system version 7.3 and above is supported. eta/DYNAFORM must run under KDE environment. The NVIDIA graphic cards are recommended.

**PC/WINDOWS**

eta/DYNAFORM in PC platform is compatible with PC-LS-DYNA 970. It will run in Windows 98, 2000, and XP environments. It is not recommended for usage with earlier versions of Windows. The following are minimum requirements for proper operation of eta/DYNAFORM in a Windows environment:

**Minimum Graphics Requirement:** XGA(1024 x 768)

**Graphics Card:** OpenGL based (Nvidia chip set recommended)

**Minimum Memory Requirement:**

Small models (10,000 - 20,000 elements): 256 megabytes RAM  
Medium models (20,000 - 50,000 elements): 384 megabytes RAM  
Large models (50,000 - 100,000 elements): 512 megabytes RAM  
Huge models (100,000 - 200,000 elements): 768+ megabytes RAM

**Minimum Load space requirement:** 256 megabytes

**Recommended processor:** Pentium 4

## APPENDIX B

## SUPPORTED IGES ENTITY TYPES

NAME	TYPE
Null Entity	0
Circular Arc Entity	100
Composite Curve Entity	102
Conic Arc Entity	104
Copious Data Entity	106
Plane Entity	108
Line Entity	110
Parametric Spline Curve Entity	112
Parametric Spline Surface Entity	114
Point Entity	116
Ruled Surface Entity	118
Surface of Revolution Entity	120
Tabulated Cylinder Entity	122
Transformation Matrix Entity	124
Rational B-Spline Curve Entity	126
Rational B-Spline Surface Entity	128
Offset Surface Entity	140
Boundary Entity	141
Curve on a Parametric Surface Entity	142
Bounded Surface Entity	143
Trimmed ( Parametric ) Surface Entity	144
Subfigure Definition Entity	308
Associativity Instance Entity	402
Property Entity	406
Singular Subfigure Instance Entity	408



## APPENDIX C

## CURVE FILE FORMAT EXAMPLE

## LIST DIRECTED FORMAT

```

RECORD 1      :N
RECORD 2      :N1 'TITLE'
RECORD 3      :XDATA1      YDATA1
.
.
.
RECORD 2 + N1 :NDATAN1      YDATAN1
RECORD 3 + N1 :N2 'TITLE FOR CURVE2'
RECORD 4 + N1 :XDATA1 YDATA1
.
.
.
etc.

```

Where:

N            = Number of curves, integer  
N1           = Number of points on the first curve, integer  
N2           = Number of points on the second curve, integer  
TITLE        = Title with a maximum of 20 characters, Starts and ends  
with a quote ('). Separate with a space between N1 or N2.

XDATA1 = X value of the first part on the curve, real number

YDATA1 = Y value of the first part on the curve, real number.  
Separate with a space between XDATA1.

etc.

## APPENDIX D

**eta/DYNAFORM SHORTCUT KEY**

Open	CTRL+O	
Import	F3	
Export	SHIFT+F3	
Exit	CTRL+X	
Create	CTRL+P	
Line/Points	CTRL+L	F4
Surface	CTRL+S	F7
Element	CTRL+E	F2
Node	CTRL+N	F6
Mesh Check & Repair	CTRL+R	F5
Boundary Condition	CTRL+U	
Nodes/Elements Set	CTRL+V	
Define Tools	CTRL+T	
Draw Bead	CTRL+D	
Define Blank	CTRL+B	
LS-DYNA	CTRL+A	
Online Help	F1	
Version	CTRL+H	

## FINAL NOTES

We at ETA would like to thank those who helped in creating this manual. We have tried to make the information in the manual as accurate as possible. In an effort to keep future versions as error free as possible, we kindly request you to send us your suggestions and notify us of any errors. You can contact the ETA software support group at our Troy headquarters via:

Voice: 248-729-3010  
Fax: 248-729-3020  
E-mail: [support@eta.com](mailto:support@eta.com)

The eta/DYNAFORM Team  
Engineering Technology Associates, Inc